



# MSC Nastran 2020

## Quick Reference Guide



HEXAGON



## **Corporate**

MSC Software Corporation  
4675 MacArthur Court, Suite 900  
Newport Beach, CA 92660  
Telephone: (714) 540-8900  
Toll Free Number: 1 855 672 7638  
[Email:](mailto:americas.contact@mscsoftware.com) americas.contact@mscsoftware.com

## **Europe, Middle East, Africa**

MSC Software GmbH  
Am Moosfeld 13  
81829 Munich, Germany  
Telephone: (49) 89 431 98 70  
[Email:](mailto:europe@mscsoftware.com) europe@mscsoftware.com

## **Japan**

MSC Software Japan Ltd.  
Shinjuku First West 8F  
23-7 Nishi Shinjuku  
1-Chome, Shinjuku-Ku  
Tokyo 160-0023, JAPAN  
Telephone: (81) (3)-6911-1200  
[Email:](mailto:MSCJ.Market@mscsoftware.com) MSCJ.Market@mscsoftware.com

## **Asia-Pacific**

MSC Software (S) Pte. Ltd.  
100 Beach Road  
#16-05 Shaw Tower  
Singapore 189702  
Telephone: 65-6272-0082  
[Email:](mailto:APAC.Contact@mscsoftware.com) APAC.Contact@mscsoftware.com

## **Worldwide Web**

[www.mscsoftware.com](http://www.mscsoftware.com)

## **Support**

<http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx>

## **Disclaimer**

MSC Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright © 2020 MSC Software Corporation. Printed in U.S.A. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC Software Corporation is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from MSC Software suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at <http://www.mscsoftware.com/thirdpartysoftware> (or successor website designated by MSC from time to time).

PCGLSS 8.0, Copyright © 1992-2016, Computational Applications and System Integration Inc. All rights reserved. PCGLSS 8.0 is licensed from Computational Applications and System Integration Inc.

MSC, Dytran, Marc, MSC Nastran, Patran, the MSC Software corporate logo, e-Xstream, Digimat, and Simulating Reality are trademarks or registered trademarks of the MSC Software Corporation and/or its subsidiaries in the United States and/or other countries. Hexagon and the Hexagon logo are trademarks or registered trademarks of Hexagon AB and/or its subsidiaries.

NASTRAN is a registered trademark of NASA. FLEXlm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. All other trademarks are the property of their respective owners.

Use, duplicate, or disclosure by the U.S. Government is subjected to restrictions as set forth in FAR 12.212 (Commercial Computer Software) and DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), as applicable.

U.S. Patent 9,361,413

April 29, 2020

NA:V2020:Z:Z:DC-QRG-PDF

# Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: [documentation-feedback@mscsoftware.com](mailto:documentation-feedback@mscsoftware.com).

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at: <http://msc-documentation.questionpro.com>.

**Note:**

The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact [Technical Support](#).



# CONTENTS

## MSC Nastran Quick Reference Guide

[Table of Contents](#)

Preface

1

[Executing MSC Nastran](#)

2

[NASTRAN Statement](#)

- List of MSC Nastran Books, xx
- Technical Support, xxi
- Training and Internet Resources, xxii
  
- Executing MSC Nastran, 2
  
- The NASTRAN Statement, 16
  - NASTRAN, 17
  - BUFFSIZE (1), 18
  - F06 (2), 18
  - NLINES (9), 18
  - MAXLINES (14), 18
  - METIME (20), 18
  - APP (21), 18
  - MACHTYPE (22), 18
  - DIAGA (25), 18
  - CONFIG (28), 18
  - MESH (31), 18
  - ADUMi (46 - 54), 18
  - HEAT (56), 18
  - DIAGB (61), 18
  - PUNCH (64), 18
  - MPYAD (66), 18
  - DCMPOUT(69), 18
  - DELFF (77), 19
  - DBSET, 19
  - DMAP (82), 19
  - F04 (86), 19
  - RADMTX (87), 19
  - RADLST (88), 19
  - SMP (107), 19
  - NEWHESS (108), 19
  - (109), 20
  - BUFFPOOL (114), 20
  - (119), 20
  - ATTDEL (124), 20
  - NOKEEP (125), 20
  - SPARSE (126), 20
  - UPDTIM (128), 20
  - SMPYAD67 (129), 20
  - MAXDBSET, 21
  - AUTOASGN (133), 21
  - TSTAMP (135), 21
  - QUADINT (141), 21
  - SCR300 (142), 21
  - LOCBULK (143), 21
  - (144), 21
  - BFGS (145), 21
  - FBSMEM (146), 22
  - UWM (147), 22
  - DBVERCHK (148), 22
  - SCR300DEL (150), 22
  - (151), 22
  - DBLAMKD (155), 22
  - (162), 23
  - (166), 23
  - LDQRKD (170), 24
  - OLDQ4K (173), 24
  - Q4TAPER (189), 24
  - Q4SKEW (190), 24
  - TETRAAR (191), 24



- SCRSAVE (196), 24
- MINFRONT (198), 24
- NSEGADD (200), 24
- CORDM (204), 24
- (205), 24
- DCMPSEQ (206), 24
- USPARSE (209), 25
- PUNCHTYPE (210), 25
- CHEXANT (212), 26
- DISTORT (213), 26
- T3SKEW (218), 26
- (219), 26
- (220), 26
- (221), 26
- (253 - 262), 26
- MAXSET (263), 26
- QUARTICDLM (270), 26
- (273), 26
- DBCFACT (274), 26
- (275), 27
- SPLINE\_METRICS (281), 27
- MINDEF (303), 27
- MPERTURB (304), 27
- (309), 27
- OLDRBE3 (310), 27
- TBCMAG (311), 27
- INDEX (316), 27
- XMSG (319), 28
- OLDDAREA (320), 28
- RSEQCONT (357), 28
- QLHOUL (359), 28
- PRTPCOMP (361), 29
- STRICTUAI (363), 29
- STPFLG (366), 29
- QRMETHOD (370), 29
- PARAMCHK (372), 29
- TZEROMAX (373), 29
- NOLIN (386), 30
- KRYLOV1 (387), 30
- KRYLOV2 (388), 30
- KRYLOV3 (389), 30
- KRYLOV4 (390), 30
- KRYLOV5 (391), 30
- BARMASS (398), 31
- DPBLKTOL (402), 31
- OP2NEW (403), 31
- DEF\_DENS (408), 31
- DEF\_TECO (410), 31
- DEF\_TEIJ (411), 31
- DEF\_DAMP (412), 31
- OPTCOD (413), 31
- (414), 32
- OLDTLDLDMTH (428), 32
- NONLRGAP (431), 32
- ESLNRO (443), 32
- (444), 32
- IFPSTAR, 32
- MNLLQ4C (445), 33
- (446), 33
- PARSLAVE(449), 33
- (451), 34
- DIFFS(614), 34
- IFPBUFF(624), 34
- NONUPIV (653), 35
- GPU\_MIN\_RANK (655), 35
- GPU\_MIN\_FRONT (656), 35
- (662), 35
- SEGLOG(664), 36
- CNTBKCMP(666), 36
- (670), 36
- LMFBKCMP(676), 36
- FF0BKCMP(677), 36
- OLDCNTR(678), 37
- (679), 37
- (684), 37
- RDBOTH(695), 37
- S2SDEF(701), 37
- HDF5 (702), 38
- OBEAMS (725), 38
- OLDCONE (726), 38
- METS2S (727), 38
- DMP (728), 38
- N2SDOF (729), 39
- H5NORDOF(730), 39
- H5MTX (739), 39
- H5MDL (740), 39
- ACCSDLSZ(747), 39
- H5GM34 (751), 39
- UDEFGRID(753), 39
- CNTBACK(758), 40
- OP2NEW(761), 40
- IGNBLN(767), 40
- BLNMEM(768), 40
- OPENFSI(776), 41
- H5INFO(789), 41
- STRNCUR(778), 41
- CNT101(786), 42
- H5XHH (790), 42

# 3

## File Management Statements

- Key to Descriptions, 44
- The File Management Section (FMS), 45
  - File Management Statement Summary, 45
  - File Management Statement Descriptions, 46
  - File Management Statements, 50
    - \$, 50
    - ACQUIRE, 51
    - ASSIGN, 52
    - CONNECT, 60
    - DBCLEAN, 64
    - DBDICT, 65
    - DBDIR, 79
    - DBFIX, 80
    - DBLOAD, 81
    - DBLOCATE, 84
    - DBSETDEL, 87
    - DBUNLOAD, 88
    - DBUPDATE, 90
    - DEFINE, 91
    - ENDJOB, 94
    - EXPAND, 95
    - INCLUDE, 97
    - INIT, 98
    - MEMLIST, 102
    - PROJ, 103
    - RESTART, 104

# 4

## Executive Control Statements

- Key to Descriptions, 108
  - Executive Control Section, 109
  - Executive Control Statement Summary, 109
- Executive Control Statement Descriptions, 110
  - \$, 111
  - ALTER, 112
  - APP, 115
  - CEND, 116
  - COMPILE, 117
  - COMPILER, 121
  - DIAG, 122
  - DOMAINsolver, 126
  - ECHO, 133
  - ENDALTER, 134
  - GEOMCHECK, 135
  - ID, 139
  - INCLUDE, 140
  - LINK, 141
  - MALTER, 143
  - MODEL\_CHECK, 147
  - SOL, 150
  - SOL 600, 153
  - SOL 700, 172
  - SPARSEsolver, 184
  - TIME, 189

# 5

## Case Control Commands

- Key to Descriptions, 192
- The Case Control Section, 193
  - Case Control Command Descriptions, 193
- Case Control Command Summary, 195
  - Subcase Definition, 195
  - Data Selection, 196
  - Output Selection, 201
  - Superelement Control, 205
  - Miscellaneous, 206
- Case Control Commands, 207
  - \$, 207
  - \$S700, 208

- A2GG (Case), 212
- ACCELERATION (Case), 213
- ACFPMRESULT (Case), 216
- ACPower (Case), 218
- ACTIVAT (Case), 219
- ACTRIM (Case), 220
- ADACT (Case), 221
- ADAMSMNF\* (Case), 222
- ADAPT (Case), 233
- AECONFIG (Case), 234
- AERCONFIG (Case), 235
- AEROF (Case), 236
- AESYMXY (Case), 237
- AESYMXZ (Case), 238
- AEUXREF (Case), 239
- ANALYSIS (Case), 240
- APRESSURE (Case), 248
- AUTOSPC (Case), 249
- AUXCASE (Case), 252
- AUXMODEL (Case), 253
- AVLEXB (Case), 254
- BC (Case), 260
- AXISYMMETRIC (Case), 261
- B2GG (Case), 262
- B2PP (Case), 263
- BC (Case), 264
- BCHANGE (Case), 265
- BCONCHK (Case), 266
- BCONTACT (Case), 267
- BCONTACT (Case), 271
- BCMOVE (Case), 274
- BEGIN BULK (Case), 275
- BENDL (Case), 278
- BOUTPUT (Case), 280
- BSQUEAL (Case), 282
- CAMPBELL (Case), 283
- CLOAD (Case), 284
- CMETHOD (Case), 285
- CMSENRGY (Case), 286
- CSSCHD (Case), 289
- DATAREC (Case), 290
- DBSAVE (Case), 291
- DEACTEL (Case), 292
- DEFORM
- DESGLB (Case), 294
- DESMOD (Case), 295
- DESOBJ (Case), 296
- DESSUB (Case), 297
- DESVAR (Case), 298
- DISPLACEMENT (Case), 299
- DIVERG (Case), 304
- DLOAD (Case), 305
- DRSPAN (Case), 306
- DSAPRT (Case), 307
- DSYM (Case), 309
- ECHO (Case), 310
- EDE (Case), 312
- EKE (Case), 315
- ELAFORCES (Case), 318
- ELSDCON (Case), 320
- ELSENS (Case), 321
- ELSUM (Case), 323
- ENDSTEP (Case), 325
- ENDTIME (Case), 326
- ENDMODULE (Case), 327
- ENTHALPY (Case), 328
- EQUILIBRIUM (Case), 329
- ERP (Case), 331
- ESE (Case), 335
- EXPORTLD (Case), 338
- EXTDRLIN (Case), 340
- EXTDROUT (Case), 349
- EXTSEOUT (Case), 351
- FATIGUE (Case), 364
- FBODYLD (Case), 372
- FEMCHECK (Case), 373
- FLSFSEL (Case), 374
- FLSPOUT (Case), 376
- FLSTCNT (Case), 379
- FLUX (Case), 381
- FMETHOD (Case), 382
- FORCE (Case), 383
- FREQUENCY (Case), 386
- FRF (Case), 387
- GPFORCE (Case), 403
- GPKE (Case), 406
- GPRSORT (Case), 407
- GPSDCON (Case), 408
- GPSTRAIN (Case), 409
- GPSTRESS (Case), 411
- GROUNDCHECK (Case), 413
- GUST (Case), 415
- GVECTOR (Case), 416
- HADAPT (Case), 417
- HARMONICS (Case), 418
- HDOT (Case), 419
- HISTOGRAM (Case), 420
- HOUTPUT (Case), 422
- HTFLOW (Case), 423
- IC (Case), 424
- ICF (Case), 426
- IMPERFECT (Case), 428
- INCLUDE (Case), 429
- INTENSITY (Case), 430

- IRLOAD (Case), 431
- K2GG (Case), 432
- K2PP (Case), 433
- K42GG (Case), 434
- LABEL (Case), 435
- LINE (Case), 436
- LOAD (Case), 437
- LOADNAME (Case), 438
- LOADSET (Case), 439
- M2GG (Case), 441
- M2PP (Case), 442
- MASSSET (Case), 443
- MASTER (Case), 444
- MAXLINES (Case), 446
- MAXMIN (Case), 447
- MAXMIN (Case), 449
- MAXMIN(DEF) (Case), 451
- MCFRACTION (Case), 454
- MCHSTAT (Case), 457
- MEFFMASS (Case), 458
- METHOD (Case), 460
- MFLUID (Case), 462
- MFREQUENCY (Case), 463
- MODALKE (Case), 467
- MODALSE (Case), 471
- MODES (Case), 475
- MODESELECT (Case), 476
- MODTRAK (Case), 482
- MONCARL (Case), 483
- MONITOR (Case), 484
- MPC (Case), 486
- MPCFORCES (Case), 487
- MPRES (Case), 490
- NLBUCK (Case), 491
- NLHARM (Case), 493
- NLIC (Case), 494
- NLLOAD (Case), 496
- NLOPRM (Case), 497
- NLPARM (Case), 501
- NLRESTART (Case), 502
- NLSTEP (Case), 505
- NLSTRESS (Case), 506
- NONLINEAR (Case), 508
- NOUTPUT (Case), 509
- NSM (Case), 510
- NVELOCITY (Case), 511
- OFREQUENCY (Case), 513
- OIMPERFECT (Case), 515
- OLOAD (Case), 516
- OMODES (Case), 519
- OTIME (Case), 521
- OUTPUT (Case), 523
- OUTRCV (Case), 524
- P2G (Case), 525
- PACCELERATION (Case), 526
- PAGE (Case), 528
- PARAM (Case), 529
- PARTN (Case), 530
- PEAKOUT (Case), 531
- PFGRID (Case), 532
- PFMODE (Case), 534
- PFPANEL (Case), 538
- PLOTID (Case), 542
- POST (Case), 543
- PRESSURE (Case), 547
- RANDOM (Case), 548
- RCROSS (Case), 549
- REPCASE (Case), 551
- RESVEC (Case), 552
- RGYRO (Case), 554
- RIGID (Case), 555
- RSDAMP (Case), 557
- SACCELERATION (Case), 558
- SDAMPING (Case), 559
- SDISPLACEMENT (Case), 561
- SEALL (Case), 562
- SEDAMP (Case), 563
- SEDR (Case), 564
- SEDV (Case), 565
- SEEINCLUDE (Case), 566
- SEFINAL (Case), 567
- SEKREDUCE (Case), 568
- SELGENERATE (Case), 569
- SELREDUCE (Case), 570
- SEMGENERATE (Case), 571
- SEMREDUCE (Case), 572
- SERESP (Case), 573
- SET (Case), 574
- SETP (Case), 576
- SETS DEFINITION (Case), 577
- SKIP (Case), 578
- SMETHOD (Case), 579
- SOLUTION (Case), 580
- SPC (Case), 581
- SPCFORCES (Case), 582
- SPLINOUT (Case), 585
- STATSUB (Case), 586
- STEP (Case), 588
- STOCHASTICS (Case), 589
- STRAIN (Case), 590
- STRESS (Case), 593
- STRFIELD (Case), 596
- SUBCASE (Case), 597
- SUBCOM (Case), 598

- SUBSEQ (Case), 600
- SUBSEQ1 (Case), 601
- SUBSTEP (Case), 602
- SUBTITLE (Case), 604
- SUPER (Case), 605
- SUPORT1 (Case), 607
- SVECTOR (Case), 608
- SVELOCITY (Case), 609
- SYM (Case), 610
- SYMCOM (Case), 611
- SYMSEQ (Case), 612
- TACCELERATION (Case), 613
- TDISPLACEMENT (Case), 615
- TEMPERATURE (Case), 617
- TERMIN (Case), 621
- TFL (Case), 622
- THERMAL (Case), 623

■ Case Control Applicability Tables, 648

■ X-Y PLOT Commands, 662

■ X-Y Output Command Summary, 662

- ALLEDGE TICS, 666
- BALL EDGE TICS, 667
- BLEFT TICS, 668
- BRIGHT TICS, 669
- CAMERA, 670
- CLEAR, 671
- CSCALE, 672
- CURVELINESYMBOL, 673
- DENSITY, 674
- LEFT TICS, 675
- LONG, 676
- LOWER TICS, 677
- PENSIZE, 678
- PLOTTER, 679
- RIGHT TICS, 680
- SEPLOT, 681
- SEUPPLOT, 682
- TALL EDGE TICS, 683
- TCURVE, 684
- TLEFT TICS, 685
- TRIGHT TICS, 686
- UPPER TICS, 687
- XAXIS, 688
- XBAXIS, 689
- XBGRID LINES, 690
- XDIVISIONS, 691
- XGRID LINES, 692
- XINTERCEPT, 693
- XLOG, 694
- XMAX, 695
- XMIN, 696
- TITLE (Case), 625
- TRIM (Case), 626
- TRIMF (Case), 627
- TRIMGRP (Case), 629
- TSTEP (Case), 630
- TSTEPNL (Case), 631
- TSTRU (Case), 632
- TVELOCITY (Case), 633
- UNGLUE (Case), 635
- VCCT (Case), 636
- VECTOR (Case), 637
- VELOCITY (Case), 638
- VINTENSITY (Case), 640
- VUGRID (Case), 643
- WEIGHTCHECK (Case), 644
- WETSENS (Case), 646
- XPAPER, 697
- XTAXIS, 698
- XTGRID LINES, 699
- XTITLE, 700
- XYPAPLOT, 701
- XYPEAK, 702
- XYPLOT, 703
- XYPRINT, 709
- XYPUNCH, 710
- XVALUE PRINT SKIP, 711
- YAXIS, 712
- YBDIVISIONS, 713
- YBINTERCEPT, 714
- YBGRID LINES, 715
- YBLOG, 716
- YBMAX, 717
- YBMIN, 718
- YTITLE, 719
- YBVALUE PRINT SKIP, 720
- YDIVISIONS, 721
- YINTERCEPT, 722
- YGRID LINES, 723
- YLOG, 724
- YMAX, 725
- YMIN, 726
- YPAPER, 727
- YTAXIS, 728
- YTDIVISIONS, 729
- YTGRID LINES, 730
- YTINTERCEPT, 731
- YTITLE, 732

# 6

## Parameters

- YTLOG, 733
- YTMAX, 734
- YTMIN, 735
- OUTPUT(POST) Commands, 739
  - SURFACE, 739
- OUTPUT(PLOT) Commands, 743
  - AXES, 745
  - CAMERA, 747
  - CONTOUR, 748
  - CSCALE, 750
  - DISTORTION, 751
  - FIND, 752
  - MAXIMUM DEFORM, 754
  - OCULAR SEPARATION, 755
  - ORIGIN, 756
  - ORTHOGRAPHIC, etc., 757
  - PAPER SIZE, 758
  - PEN, 759
- YTITLE, 736
- YTVALUE PRINT SKIP, 737
- YVALUE PRINT SKIP, 738
- VOLUME, 742
- PERSPECTIVE, 760
- PLOT, 761
- PLOTTER, 769
- PROJECTION, 770
- PTITLE, 771
- SCALE, 772
- SEPLOT, 773
- SET, 774
- SEUPPLOT, 777
- STEREOSCOPIC, 778
- VANTAGE POINT, 779
- VIEW, 780

## ■ Parameter Descriptions, 784

- ACEXTMD, 784
- ACEXTSET, 784
- ACOUT, 784
- ACOWEAK, 784
- ACSYM, 785
- ACTDMP, 785
- ACTSMP, 785
- ACTMEM, 785
- ADJMETH, 785
- ADMEXTU, 786
- ADMPOST, 786
- ADPCON, 786
- ADSTAT, 786
- AERODOF, 787
- AESDISC, 787
- AESMAXIT, 787
- AESMETH, 787
- AESRNDM, 787
- AESTOL, 787
- AGGRROT, 788
- ALPHA1, ALPHA2, 788
- ALPHA1FL, ALPHA2FL, 789
- ALTRED, 790
- ALTSHAPE, 790
- ARBMASP, 790
- ARBMFEM, 790
- ARBMNOW, 790
- ARBMPS, 791
- ARBMSS, 791
- ARBMSTYP, 791
- ARF, 791
- ARS, 791
- ASCOUP, 791
- , 791
- ASING, 792
- AUNITS, 792
- AUTOADJ, 792
- AUTOGOUT, 793
- AUTOMSET, 793
- AUTOQSET, 794
- AUTOSPC, 795
- AUTOSPCR, 795
- AUTOSPRT, 795
- BAILOUT, 795
- BEAMBEA, 795
- BEIGRED, 796
- BIGER, BIGER1, BIGER2, BIGER3, 796
- BUCKLE, 796
- CASIEMA, 796
- CASIMEST, 797
- CASPIV, 797
- CB1, CB2, 797
- CDIF, 797
- CDITER, 797
- CDPCH, 798
- CDPRT, 798
- CFDIAGP, 798
- CFRANDEL, 798

- CHECKOUT, 798
- CK1, CK2, 799
- CK3, 800
- CLOSE, 800
- CM1, CM2, 800
- COMPMATT, 800
- CONFAC, 801
- COSUBCYC, 801
- COSUBMAX, 801
- COUPMASS, 802
- CP1, CP2, 802
- CQC, 802
- CURV, 803
- CURVPLOT, 805
- CWDIAGP, 806
- CWRANDEL, 806
- DBALL, 806
- DBCCONV, 807
- DBCDIAG, 807
- DBCOWRT, 807
- DBDICT, 807
- DBDN, 808
- DBDRPRJ, 808
- DBDRVER, 808
- DBEXT, 808
- DBRCV, 808
- DBUP, 808
- DDRMM, 808
- DELCLUMP, 808
- DESPCH, 809
- DESPCH1, 809
- DFREQ, 810
- DIROUT, 810
- DOPT, 810
- DPEPS, 810
- DPHFLG, 811
- DSNOKD, 811
- DSO, 812
- DSZERO, 812
- DV3PASS, 812
- DYNSPCF, 812
- EIGFILT, 812
- ENFMETH, 812
- ENFMOTN, 813
- EPPRT, 814
- EPSILONT, 814
- EPZERO, 815
- ERPC, 815
- ERPREFDB, 815
- ERPRHO, 815
- ERPRLF, 815
- ERROR, 815
- ESLFSAV, 815
- ESLMOVE, 816
- ESLLCOMP, 816
- ESLMPC1, 816
- ESLOPTEX, 816
- ESLPRT, 817
- ESLPRT1, 817
- ESLRCF,user\_rc\_file, 817
- ESLTOPCV, 818
- ESLUNT2, 818
- ESLUNT1, 818
- EST, 818
- EULBND, 818
- EULSTRES, 819
- EUSUBCYC, 820
- EXCLUDE, 820
- EXTDR , 821
- EXTDROUT , 821
- EXTDRUNT, 821
- EXTOUT, 821
- EXTRCV, 824
- EXTUNIT, 824
- , 824
- FACTOR, 825
- FASTFR, 825
- FBATOLR, 825
- FBLEND, 825
- FDRLDS, 826
- FIXEDB, 827
- FKSYMFAC, 827
- FLEXINCR, 828
- FLUIDMP, 828
- FLUIDNE, 828
- FLUIDSE, 829
- FMULTI, 829
- FOLLOWK, 829
- FRQDEPO, 829
- FULLSEDR, 830
- FZERO, 830
- G, GFL, 830
- GEOMU, 830
- GPECT, 830
- GRADMESH, 830
- GRAVSET, 831
- GRDPNT, 831
- GUSTAERO, 832
- GYROAVG, 832
- HEATCMD, 832
- HEATSTAT, 833
- HFREQ, HFREQFL, 833
- HTOCITS, 833
- HTOCPRT, 834

- HTOCTOL, 834
- HTSYM, 834
- ICOPT, 834
- IFP, 834
- INREL, 834
- INRLM, 835
- IRES, 835
- ITAPE, 835
- IUNIT, 836
- JWLDLT, 836
- KDAMP, KDAMPFL, 836
- KDIAG, 836
- KDMFILT, 837
- K4RITZ, 838
- K6ROT, 838
- LANGLE, 839
- LDSUM, 839
- LFREQ, LFREQFL, 840
- LGDISP, 840
- LMFAC, 840
- LMODES, LMODESFL, 841
- LOADU, 841
- LOOPID, 841
- LSTRN, 842
- , 842
- MACH, 842
- MARALPHA, 842
- MARAUTOC, 843
- MARBATCH, 843
- MARBK105, 844
- MARBK106, 844
- MARC4401, 844
- MARC7601, 844
- MARCASUM, 845
- MARCAUTO, 845
- MARCAXEL, 846
- MARCBEAM, 846
- MARCBODY, 847
- MARCBUSH, 847
- MARCCBAR, 847
- MARCCENT, 848
- MARCCON2, 848
- MARCCON3, 848
- MARCCPY, 848
- MARCDEF, 849
- MARCDILT, 850
- MARCDIS2, 850
- MARCDIS3, 850
- MARCDIS4, 850
- MARCDMIG,N, 851
- MARCDUPE, 851
- MARCDYND, 851
- MARCEKND, 852
- MARCEXIT, 852
- MARCFEAT,N, 852
- MARCFILI, 852
- MARCFRIC, 853
- MARCGAPD, D, 853
- MARCGAPN, ID, 854
- MARCGAPP, 854
- MARCGAUS, 854
- MARCGLUE, 854
- MARCGRAV, 855
- MARCHOST, 855
- MARCIAMN, 855
- MARCINTC, 856
- MARCINTF, 856
- MARCITER, 856
- MARCL001, 857
- MARCLOWE, 858
- MARCLUMP, 858
- MARCMAT2, 858
- MARCMAT3, 858
- MARCMATT, 859
- MARCMEM, Value, 859
- MARCMID3, 860
- MARCMNF, 860
- MARCMPCC, 860
- MARCMPII, 861
- MARCND99, 861
- MARCNOER, 861
- MARCOFFT, 862
- MARCONTF, 862
- MARCOOCC, 862
- MARCOPT, 863
- MARCOSET, 863
- MARCOTIM, 864
- MARCOUTR, 864
- MARCPARR, 864
- MARCPENT, 865
- MARCPINN, 865
- MARCPLAS, n, 865
- MARCPOS, 866
- MARCPOST, 866
- MARCPR99, 867
- MARCPRN, 867
- MARCPRNG, 867
- MARCPRNR, 867
- MARCPROG, 868
- MARCRACC, 868
- MARCRBAL, 869
- MARCRBAR, 869
- MARCRBE2, 869
- MARCRBE3, 870

- MARCREVR, 870
- MARCREVRX, 870
- MARCRIGD, 871
- MARCSAME, 871
- MARCSCLR, 872
- MARCSETS, 872
- MARCSETT, 873
- MARCSINC, 873
- MARCSIZ3, Value, 873
- MARCSIZ4, Value, 873
- MARCSIZ5, Value, 873
- MARCSIZ6, Value, 874
- MARCSLHT, 874
- MARCSOLV, 874
- MARCSPCC, 875
- MARCSTIFF, Time, 875
- MARCSTOP, 875
- MARCSUMY, 875
- MARCT16, 876
- MARCT19, 877
- MARCTABL, 877
- MARCTEDF, 877
- MARCTEDN, 878
- MARCTEMP, 878
- MARCTETT, 878
- MARCTIEC, 879
- MARCTOL, 879
- MARCTOD, 879
- MARCTOTL, 879
- MARCTOTT, 880
- MARCTUBE, 880
- MARCTVL, Value, 880
- MARCUSUB, chr, 881
- MARCVERS, 881
- MARCWDIS, 881
- MARCWELD, 882
- MARELSTO, 882
- MARGPFEL, 882
- MARFACEA, 882
- MARFACEB, 883
- MARFATAL, 883
- MARGPFOR, 883
- MARHEATM, 883
- MARHTPRT, 884
- MARIBOOC, 884
- MARIPROJ, 884
- MARLDCMB, 884
- MARLDRMV, 885
- MARMPCHK, 885
- MARMPCID, 886
- MARMTLCK, 886
- MARNOCID, 887
- MARNOSSET, Name, 887
- MAROFSET, 887
- MARPLANE, 887
- MARPROCS, 888
- MARRBAR2, 888
- MARROUTT, 888
- MARSHRII, 889
- MARTET10, 889
- MARTEPIN, 890
- MARUPDAT, 890
- MARVFCUT, 891
- MAUTOSPC, 891
- MAXAPL, 891
- MAXDAMP, 891
- MAXIREV, 892
- MAXLP, 893
- MAXRATIO, 893
- MBENDCAP, 893
- MCNLPARM, 894
- MCON2D3D, 894
- MCORDUPD, 894
- MCSHLCK, 894
- MCSHORR, 895
- MDAREAMD, 896
- MDEFSEPP, 896
- MDK4OPT, 897
- MDOPT14, 897
- MDOTM, 897
- MDOTMFAC, 897
- MDREDOPT, 897
- MDUMLOAD, 898
- MECHFIL, 898
- MECHFIX, 898
- MECHPRT, 898
- MESH, 898
- METHCMRS, 899
- MEXTRNOD, 899
- MEXTSEE,N, 899
- MFASTCMP, 899
- MFEA5701, 900
- MFORCOR1, 900
- MFORDUPE, 900
- MFSKIPPP, 901
- MGAPINIT, 901
- MGLUETOL, 901
- MHEATSHL, 901
- MHEATUNT, 902
- MHEMIPIX, 902
- MHOUBOLT, 902
- MHRED, 902
- MICRO , 903
- MINCLDSB, 903

- MINIGOA, 903
- MINRECCC, N, 904
- MINSOUTT, 904
- MINVASHF, 904
- MINVCITR, 904
- MINVCSHF, 904
- MINVCTOL, 904
- MINVFMAX, 905
- MINVNMOD, 905
- MLDALLOW, 905
- MLSTRAIN, 905
- MLSTRAN2, 906
- MMAT2ANI, 906
- MMBOLTUS, 907
- MMEMDETT, 907
- MMFIL, 908
- MMMETH, 908
- MNASTLDS, 908
- MODACC, 909
- MODEL, 909
- MOFFCORE, 910
- MOUTRCOR, 910
- MP1SET, 910
- MPCX, 911
- MPERMPRT, 911
- MPTUNIT, 911
- MQUATERN, 911
- , 911
- MRADUNIT, 912
- MRAFFLOR, N, 912
- MRAFFLOT, N, 912
- MRAFFLOW, Name, 912
- MRALIAS ID (MALIAS02, MALIAS03, etc.), 913
- MRALLOGC, 913
- MRALLOCS, 913
- MRBE3SNG, 914
- MRBEAMB, 914
- MRBEPARM, IJK, 914
- MRBDYCVT, 915
- MRBIGMEM, 915
- MRBUKMTH, 915
- Default = 2, SOL 600 only, 915
- MRC2DADD, 916
- MRCKBODY, 916
- MRCKLOAD, 916
- MRCNADD, 917
- MRCNRES, 917
- MRCNTAB, 917
- MRCNUSE, 918
- MRCONVER, 918
- MRCORDS, 918
- MRCQUAD4, 919
- MRCTRIA3, 919
- MRCWANGL, 919
- MRDELTTT, 919
- MRDISCMB, 920
- MRDUPMAT, 921
- MRDYNOLD, 921
- MREIGMTH, 921
- MREL1103, 922
- MRELRB, 922
- MRELSPCD, 922
- MRENUELE, 923
- MRENUGRD, 923
- MRENUMBR, 924
- MRESTALL, 924
- MRESULTS, 924
- MREVPLST, 925
- MRFINITE, 925
- MRFOLLOW, 925
- MRFOLLO2, 925
- MRFOLLOW1, 925
- MRFOLLOW3, 926
- MRFOLLOW4, 926
- MRGAPUSE, 926
- MRHERRMN, 927
- MRHYPMID, 927
- MRITTYPE,, 927
- MRMAT8A3, Value, 928
- MRMAT8E3, Value, 928
- MRMAT8N1, Value, 928
- MRMAT8N3, Value, 928
- MRMAXISZ, 928
- MRMAXNUM, 928
- MRMEMSUM, 928
- MRMTXKGG, Name, 929
- MRMTXNAM, Name, 929
- MRNOCOMP, 929
- MRNOCOR, 930
- MRNOECHO, 930
- MRORINTS, 930
- MROUTLAY, 931
- MRPARALL, 932
- MRPBUSHT, 932
- MRPIDCHK, 933
- MRPELAST, 933
- MRPLLOAD4, 933
- MRPLOD4R, 934
- MRPLSUPD, 934
- MRPOISCK, 934
- MRPREFER, 935
- MRPRSFAC, 935
- MRPSHCMP, 936

- MRPSSHELL, 936
- MRRBE3TR, 937
- MRRCFILE, RCF, 937
- MRRELNOD, 938
- MRRSTOP2, 938
- MRSCMOD, 938
- MRSETNA1, N, 939
- MRSETNA2, M, 939
- MRSETNAM, N, 939
- MRSPAWN2, CMD, 940
- MRSPLINE, 940
- MRSPRING, 940
- MRSRVER, 941
- MRSTEADY, 941
- MRT16STP, N, 941
- MRTABLIM, 941
- MRTABLS1, 942
- MRTABLS2, 943
- MRTABPRC, 943
- MRTFINAL, 943
- MRTTHREAD, 943
- MRTIMING, 944
- MRTSHEAR, 944
- MRUSDATA, 944
- MRVFIMPL, 945
- MRV09V11, 945
- MSIZOVRD, 945
- MSOLMEM, MBYTE, 946
- MSPCCCHK, 946
- MSPCKEEP, 946
- MSPEEDCB, 947
- MSPEEDCW, 947
- MSPEEDOU, 947
- MSPEEDP4, 947
- MSPEEDPS, 948
- MSPEEDSE, 948
- MSPEEDS2, 948
- MSPEEDSH, 949
- MSPLINCO, 949
- MSTFBEAM, 949
- MSTRFREE, 949
- MSTTDYNE, 950
- MTABIGNR, 950
- MTABLD1M, 951
- MTABLD1T, 951
- MTEMPCHK, 951
- MTEMPDWN, 952
- MTET4HYP, 952
- MTLD1SID, 953
- MUALLUDS, 953
- MULRFORC, 953
- MUMPSOLV, 953
- MUSBKEEP, 954
- MVERMOON, 954
- MWINQUOT, 954
- MWLDSGA, 954
- MXICODEO, 955
- MXLAGM1, 955
- NASPRT, 955
- NDAMP, NDAMPM, 956
- NEWMARK, 957
- NFDOPTS, 957
- NHPLUS, 957
- NIINTPTS, 957
- NLAYERS, 957
- NLHTLS, 958
- NLHTOL, 958
- NLHTWK, 958
- NLMAX, 958
- NLMIN, 958
- NLPACK, 958
- NLTOL, 959
- NMLOOP, 960
- NMNLFRQ, 960
- NOCOMP, 960
- NODCMP, 960
- , 960
- , 960
- , 960
- NOELOF, 961
- NOELOP, 961
- NOGPF, 961
- NOMSGSTR, 961
- NONCUP, 961
- NQSET, 961
- NUMOUT, NUMOUT1, NUMOUT2, NUMOUT3, 962
- OELMOPT, 962
- OELMSET, 962
- OG, 962
- OGEOM, 962
- OGRDOPT, 962
- OGRDSET, 963
- OLDSEQ, 963
- OLDWELD, 965
- OMACHPR, 965
- OMAXR, 965
- OMID, 965
- OMSGLVL, 965
- OPCHSET, 966
- OPGEOM, 966
- OP2GM34, 966
- OPGTKG, 966
- OPPHIB, 966

- OPPHIPA, 966
- OPTEXIT, 966
- OPTION, 967
- OSETELE, 967
- OSETGRD, 967
- OSWELM, 968
- OSWPPT, 968
- OUGCORD, 968
- OUGSPLIT, 968
- OUNIT1, 969
- OUNIT2, 969
- OUTOPT, 969
- PACINFIID, 969
- PANELMP, 969
- PATPLUS, 969
- PATVER, 969
- PCOMPRM, 969
- PDRMSG, 970
- PEDGEP, 970
- PEMFRIM, 970
- PEMMAPC, 970
- PEMNCOP4, 971
- PEMNPART, 971
- PEMSKIN, 971
- PENFN, 971
- PERCENT, 971
- PH2OUT, 972
- PKRSP, 972
- PLTMSG, 972
- POST, 972
- PARAM,POST,-7, 978
- POSTEXT, 978
- POSTADF, 979
- POSTU, 979
- PREFDB, 979
- PRGPST, 979
- PRINT, 979
- PROUT, 979
- PRPA , 979
- PRPJ, 979
- PRPHIVZ, 980
- PRTMAXIM, 980
- PRTRESLT, 980
- PSENPCB, 980
- PVALINIT, 980
- Q, 980
- RADMOD, 981
- RBSETPRT, 981
- RESLOPT, 981
- RESVEC, 981
- RFORSET, 981
- RHOCB, 981
- RIMINTP, 982
- RKSCHEME , 982
- , 982
- RMSINT, 982
- ROHYDRO , 983
- ROMULTI , 983
- ROSTR , 984
- ROTSEKE, 984
- RSPECTRA, 985
- RSPRINT, 985
- RSTTEMP, 985
- S1, S1G, S1M, 985
- S1AG,S1AM, 987
- SBSPFORM, 987
- SCRSPEC, 987
- SDCSV, 988
- SDRPOPT, 989
- SDUNIT,n, 989
- SEKD, 989
- SEMAP, SEMAPOPT, SEMAPPRT, 989
- SENSUOO, 991
- SEP1XOVR, 991
- SEQOUT, 991
- SERST, 991
- SESDAMP, 992
- SESEF, 992
- SHIFT1, 992
- SHLDAMP, 993
- SIGMA, 993
- SKINOUT, 993
- SKPAMP, 993
- SLOOPID, 993
- SMALLQ, 994
- SNORM, 994
- SNORMPRT, 995
- SOFTEXIT, 995
- SOLADJC, 995
- SOLID, 996
- SPARSEDIM, 996
- SPARSEDRI, 996
- SPARSEPH, 997
- SPDRRAT, 997
- SPCGEN, 997
- SPDMRAT, 997
- SQSETID, 997
- SRCOMPS, 997
- SRTELTYP, 998
- SRTOPT, 998
- START, 998
- STEPFC, 998
- STEPFCCTL, 998
- STIME, 998

- STRUCTMP, 999
  - SUBCASID, 999
  - SUBID, 999
  - SUPAERO, 999
  - SUPDOF, 999
  - SUPER, 1000
  - TABID, 1000
  - TABS, 1000
  - TDAMP, 1000
  - TESTNEG, 1000
  - TFSYMFAC, 1001
  - TINY, 1001
  - TOLRSC, 1001
  - TOPOCONV, 1002
  - TRMBIM, 1002
  - TSTATIC, 1002
  - UGASC, 1002
  - UNSYMF, 1003
  - UNSYMKB, 1003
  - UPDTBSH, 1003
  - USETPRT, 1003
  - USETSEL, 1004
  - USETSTRi, 1004
  - VARPHI, 1005
  - VELCUT, 1005
  - VMOPT, 1006
  - VREF, 1006
  - VUELJUMP, VUGJUMP, 1006
  - VUBEAM,  
    VUHEXA, VUPENTA, VUQUAD4, VUTETR  
    A, VUTRIA3, 1006
  - WHIRLOPT, 1007
  - WMODAL, 1007
  - WRBEAMB, 1007
  - WTMAS600, 1008
  - WTMASS, 1008
  - W3, W4, W3FL, W4FL, 1008
  - WR3, WR4, WRH, 1009
  - XFACT, 1009
  - XFACTX, 1009
  - XFLAG, 1009
  - XYMPCH, 1010
  - XYUNIT, n, 1011
  - ZROCMAS, 1011
  - ZROVEC, 1011
- Parameter Applicability Tables, 1012

## ■ Item Code Description, 1046

## ■ Element Stress (or Strain) Item Codes, 1047

- CAXIF2 (47), 1048
- CAXIF3 (48), 1048
- CAXIF4 (49), 1048
- CAXISYM (241), 1049
- CBAR (34), 1049
- CBAR (100), 1050
- CBAR (238), 1051
- CBEAM (2), 1051
- CBEAM (94), 1051
- CBEAM (239), 1052
- CBEAM3 (184), 1052
- CBEND (69), 1053
- CBUSH (102), 1053
- CBUSH1D (40), 1054
- CCONEAX (35), 1054
- CDUM3, 1055
- CDUM9 (55-61), 1055
- CELAS1 (11), 1055
- CELAS2 (12), 1055
- CELAS3 (13), 1055
- CGAP (86), 1055
- CHEXA (67), 1055
- CHEXA (93), 1056
- CHEXA FD (202), 1057
- CHEXA FD (207), 1057
- CIFHEX (65), 1057
- CIPENT (66), 1058
- CIFQDX (73), 1058
- CIFQUAD (63), 1058
- CONROD (10), 1059
- CONROD (92), 1059
- CPENTA (68), 1059
- CPENTA (91), 1060
- CPENTA FD (204), 1061
- CPENTA FD (209), 1061
- CPYRAM (242), 1062
- CPYRAM (242), 1063
- CQUAD4 (33), 1063
- CQUAD4 (90), 1064
- CQUAD42 (95), 1064
- CQUAD4 (144), 1065
- CQUAD8 (64), 1066
- CQUAD82 (96), 1066
- CQUAD FD (201), 1067

- CQUADFD (208), 1067
- CQUADR (82), 1067
- CQUADR (172), 1067
- CQUADR (232), 1067
- CQUADX(18), 1068
- CQUADXFD (214), 1068
- CQUADXFD (215), 1069
- CROD (1), 1069
- CROD (89), 1069
- CSHEAR (4), 1069
- CSLOT3 (50), 1069
- CSLOT4 (51), 1069
- CTETRA (39), 1070
- CTETRA (85), 1071
- CTETRAFD (205), 1071
- CTETRAFD (210), 1071
- CTRIA3 (74), 1072
- CTRIA32 (97), 1072
- CTRIA3 (88), 1072
- CTRIA6 (75), 1072
- CTRIA62 (98), 1072
- CTRIAFD (206), 1072
- CTRIAFD (211), 1073
- CTRIAR (70), 1073
- CTRIAR (173), 1073
- CTRIAR (233), 1073
- CTRIAFD (17), 1073
- CTRIAFD (53), 1073
- CTRIAFD (212), 1074
- CTRIAFD (213), 1074
- CTUBE (3), 1074
- CTUBE (87), 1074
- CWELDP, 1075
- (118), 1075
- CWELDC, 1075
- (117), 1075
- CWELD, 1075
- (200), 1075
- VUHEXA (145), 1075
- VUPENTA (146), 1075
- VUTETRA (147), 1075
- VUHEXA (145), 1076
- VUPENTA (146), 1076
- VUTETRA (147), 1076
- VUQUAD (189)  
VUTRIA (190), 1077
- VUQUAD (189)  
VUTRIA (190), 1078
- VUQUAD (189)  
VUTRIA (190), 1080
- VUQUAD (189)  
VUTRIA (190), 1081
- VUQUAD (189)  
VUTRIA (190), 1082
- VUBEAM (191), 1083

■ Element Force Item Codes, 1085

- CBAR (34), 1085
- CBAR (100), 1085
- CBEAM (2), 1086
- CBEAM3 (184), 1086
- CBEND (69), 1087
- CBUSH (102), 1088
- CCONEAX  
(35), 1088
- CDAMP1 (20), 1088
- CDAMP2 (21), 1088
- CDAMP3 (22), 1088
- CDAMP4 (23), 1088
- CDUM3 thru CDUM9 (55 - 61), 1088
- CELAS1 (11), 1089
- CELAS2 (12), 1089
- CELAS3 (13), 1089
- CELAS4 (14), 1089
- CGAP (38), 1089
- CONROD (10), 1089
- CQUAD4 (33), 1089
- CQUAD4 (95), 1090
- CQUAD4 (144), 1090
- CQUAD8 (64), 1091
- CQUAD82 (96), 1092
- CQUADR (82), 1092
- CQUADR (235), 1092
- CROD (1), 1093
- CSHEAR (4), 1093
- CTRIA3 (74), 1094
- CTRIA32 (97), 1094
- CTRIA6 (75), 1094
- CTRIA62 (98), 1094
- CTRIAR (70), 1094
- CTRIAR (236), 1095
- CTUBE (3), 1095
- CVISC (24), 1095
- CWELDP (118), 1095
- CWELDC (117), 1095
- CWELD (200), 1095
- VUQUAD (189)  
VUTRIA (190), 1095
- VUQUAD (189)  
VUTRIA (190), 1096
- VUBEAM (191), 1097

- Fluid Virtual Mass Pressure Item Codes, 1099
- 2D Slideline and 3D Surface Contact Item Codes, 1100
  - CSLIFID (116), 1100
  - 203, 1101
- Element Strain Energy Item Codes, 1103
- Fatigue Item Codes, 1104
- Equivalent Radiated Power (ERP) Item Codes, 1110

## 8

### Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions, 1112
- Degree-of-Freedom Set Bulk Data Entries, 1116

## 9

### Bulk Data Entries

- Key to Descriptions, 1118
  - The Bulk Data Section, 1119
- Bulk Data Entry Descriptions, 1120
  - Format of Bulk Data Entries, 1121
  - Continuations, 1126
  - Bulk Data Entry Summary, 1134
  - Constraints and Partitioning, 1135
  - Elements, 1137
  - Geometry, 1146
  - Loads, 1150
  - Materials, 1154
  - Miscellaneous, 1156
  - Nastran Implicit Nonlinear (SOL 600), 1162
  - Explicit Nonlinear (SOL 700), 1165
  - Solution Control, 1176
- - \$, 1181
  - /, 1182
  - ABINFL, 1186
  - ABSNMVB, 1188
  - ABSNMVI, 1189
  - ABSNMVR, 1190
  - ABSNMVS, 1191
  - ABSTRCT, 1193
  - ACCEL, 1194
  - ACCEL1, 1196
  - ACCSSPT, 1197
  - ACLOAD, 1198
  - ACMODL, 1199
  - ACPEMCP, 1205
  - ACSRCE, 1208
  - ACTIVAT, 1210
  - ACTRIM, 1211
  - ADAPT, 1212
  - ADUMi, 1215
  - AECOMP, 1216
  - AECOMPL, 1217
  - AEDW, 1218
  - AEFACT, 1219
  - AEFORCE, 1220
  - AEGRID, 1221
  - AELINK, 1222
  - AELIST, 1223
  - AELISTC, 1224
  - AEPARM, 1225
  - AEPRESS, 1226
  - AEQUAD4, 1227
  - AERO, 1228
  - AEROS, 1229
  - AESCALE, 1230

- AESTAT, 1231
- AESURF, 1233
- AESURFS, 1235
- AETRIAS3, 1236
- ALIASM, 1237
- ASET, 1239
- ASET1, 1240
- ATBACC, 1241
- ATBJNT, 1242
- ATBSEG, 1244
- AXIC, 1245
- AXIF, 1247
- AXSLOT, 1249
- BAROR, 1250
- BARRIER, 1252
- BCAUTOP, 1254
- BCBDPBP, 1257
- BCBMRAD, 1263
- BCBODY, 1265
- BCBODY, 1277
- BCBODY1, 1292
- BCBOX, 1294
- BCBZIER, 1296
- BCELIPS, 1298
- BCGRID (SOL 700 only), 1299
- BCGRID, 1300
- BCHANGE, 1301
- BCHANGE, 1303
- BCMATL, 1305
- BCMOVE, 1306
- BCMOVE, 1307
- BCNURB2, 1309
- BCNURBS, 1311
- BCNECT, 1313
- BCONP, 1315
- BCONPRG, 1317
- BCONPRG-700, 1326
- BCONPPR, 1331
- BCONPPR-700, 1336
- BCONUDS, 1338
- BCONUDS, 1340
- BCPARA, 1342
- BCPARA, 1349
- BCPATCH, 1355
- BCPFLG, 1356
- BCPROP, 1359
- BCPROP, 1360
- BCRIGID, 1361
- BCRGSRF, 1363
- BCSCAP, 1364
- BCSEG, 1366
- BCSURF, 1367
- BCTABLE, 1377
- BCTABLE - Glued Option, 1392
- BCTABLE, 1396
- BCTABL1, 1403
- BCTRIM, 1405
- BDYLIST, 1406
- BDYOR, 1407
- BEADVAR, 1408
- BEAMOR, 1412
- BFRIC, 1414
- BIAS, 1415
- BJOIN, 1418
- BLSEG, 1420
- BNDFIX, 1422
- BNDFIX1, 1423
- BNDFREE, 1425
- BNDFRE1, 1426
- BNDGRID, 1428
- BOLT, 1429
- BOUTPUT, 1431
- BRKSQL, 1433
- BSET, 1436
- BSET1, 1437
- BSQUEAL, 1439
- BSURF, 1445
- BWIDTH, 1447

- CAABSF, 1449
- CACINF3, 1451
- CACINF4, 1452
- CAERO1, 1453
- CAERO2, 1455
- CAERO3, 1457
- CAERO4, 1459
- CAERO5, 1461
- CAMPBLL, 1465
- CAXIFI, 1467
- CAXISYM, 1468

- CBAR, 1469
- CBARAO, 1474
- CBEAM, 1476
- CBEAM3, 1481
- CBEND, 1485
- CBUSH, 1488
- CBUSH1D, 1492
- CBUSH2D, 1493
- CCONEAX, 1494
- CDAMP1, 1495
- CDAMP1D, 1496

- CDAMP2, 1497
- CDAMP2D, 1498
- CDAMP3, 1499
- CDAMP4, 1500
- CDAMP5, 1501
- CDUMi, 1502
- CELAS1, 1503
- CELAS1D, 1504
- CELAS2, 1506
- CELAS2D, 1507
- CELAS3, 1509
- CELAS4, 1510
- CFAST, 1511
- CFLUIDi, 1513
- CGAP, 1515
- CHACAB, 1517
- CHACBR, 1519
- CHBDYE, 1521

■ , 1557

- COHESIV, 1557
- COHFRIC, 1559
- COMPUDS, 1560
- CONCTL, 1563
- CONM1, 1564
- CONM2, 1565
- CONROD, 1567
- CONTRLT, 1569
- CONV, 1573
- CONVM, 1575
- CORD1C, 1578
- CORD1R, 1580
- CORD1S, 1581
- CORD2C, 1583
- CORD2R, 1585
- CORD2S, 1587
- CORD3G, 1589
- CORD3R, 1591
- COUCOHF, 1592
- COUOPT, 1593
- COUP1FL, 1595
- COUPINT, 1596
- COUPLE, 1597
- CPENTA, 1601
- CPYRAM, 1605
- CQUAD, 1607
- CQUAD4, 1609
- CQUAD8, 1613
- CQUADR, 1617
- CQUADX, 1620
- CRAC2D, 1622
- CHBDYG, 1525
- CHBDYP, 1529
- CHEXA, 1533
- CIFHEX, 1537
- CIFPENT, 1539
- CIFQDX, 1541
- CIFQUAD, 1543
- CINTC, 1545
- CLOAD, 1546
- CMARKB2, 1547
- CMARKN1, 1548
- CMASS1, 1549
- CMASS2, 1550
- CMASS3, 1551
- CMASS4, 1552
- CMREBAI, 1553
- CMREBAR, 1555
- CRAC3D, 1624
- CREEP, 1628
- CROD, 1633
- CSEAM, 1635
- CSET, 1644
- CSET1, 1645
- CSHEAR, 1647
- CSLOT3, 1650
- CSLOT4, 1651
- CSPR, 1652
- CSSCHD, 1653
- CSSHL, 1654
- CSSHLH, 1656
- CSSHLM, 1657
- CSSHLR, 1658
- CSUPER, 1659
- CSUPEXT, 1661
- CTETRA, 1662
- CTRIA3, 1665
- CTRIA6, 1669
- CTRIAR, 1673
- CTRIAZ, 1676
- CTRIAZ6, 1678
- CTUBE, 1682
- CVISC, 1683
- CWELD, 1684
- CYAX, 1694
- CYJOIN, 1695
- CYLINDR, 1697
- CYSUP, 1698
- CYSYM, 1700

- DAMPING, 1701
- DAREA, 1704
- DCONADD, 1705
- DCONSTR, 1706
- DDVAL, 1708
- DEACTEL, 1710
- DEACTEL, 1711
- DEFORM, 1712
- DEFUSET, 1713
- DELAY, 1714
- DEQATN, 1715
- DESVAR, 1720
- DETSPH, 1721
- DIVERG, 1722
- DLINK, 1723
- DLOAD, 1724
- DMI, 1725
- DMIAx, 1729
- DMIG, 1731
- DMIG,UACCEL, 1734
- DMIGOUT, 1736
- DMIGROT, 1740
- DMIJ, 1742
- DMJi, 1745
- DMIK, 1748
- DOPTPRM, 1751
- DPHASE, 1759
- DRESP1, 1760
- DRESP2, 1772
- DRESP3, 1778
- DSCREEN, 1782
- DTABLE, 1783
- DTABLE2, 1784
- DTI, 1785
- DTI,ESTDATA, 1787
- DTI,INDTA, 1791
- DTI,SETREE, 1794
- DTI,SPECSEL, 1796
- DTI,SPSEL, 1797
- DTI,UNITS, 1798
- DVBSHAP, 1800
- DVCREL1, 1801
- DVCREL2, 1803
- DVGRID, 1805
- DVLREL1, 1807
- DVMREL1, 1809
- DVMREL2, 1811
- DVPREL1, 1813
- DVPREL2, 1815
- DVSHAP, 1817
- DVPSURF, 1818
- DYFSISW, 1819
- DYPARAM, ATBAOUT, 1820
- DYPARAM, ATBHOUT, 1821
- DYPARAM, ATBTOUT, 1822
- DYPARAM,AUTOCOUP, 1823
- DYPARAM,AXIALSYM, 1826
- DYPARAM,AXREMAP, 1828
- DYPARAM,BULKL, 1829
- DYPARAM,BULKQ, 1830
- DYPARAM,BULKtyp, 1831
- DYPARAM,CFULLRIG, 1832
- DYPARAM,CLUFLIM, 1833
- DYPARAM,CLUMPENR, 1834
- DYPARAM,CONM2OUT, 1835
- DYPARAM,CONTACT, 1836
- DYPARAM,COSUBMXT, 1839
- DYPARAM,COUFRIC, 1840
- DYPARAM,COHESION, 1841
- DYPARAM,DUMSEGS, 1842
- DYPARAM,ELDLTH, 1843
- - SOL700, 1843
- DYPARAM,EULERCB, 1844
- DYPARAM,EULERPR, 1845
- DYPARAM,EULTRAN, 1847
- DYPARAM,EUSUBMAX, 1848
- DYPARAM,EUSUBCYC, 1850
- DYPARAM,FAILDT, 1851
- DYPARAM,FAILOUT, 1852
- DYPARAM,FLOWMETH, 1853
- DYPARAM,GEOCHECK, 1856
- DYPARAM,FASTCOUP, 1857
- DYPARAM,HYDROBOD, 1859
- DYPARAM,HGCMEM, 1860
- DYPARAM,HGCOEFF, 1861
- DYPARAM,HGCSOL, 1862
- DYPARAM,HGCTWS, 1863
- DYPARAM,HGCWRP, 1864
- DYPARAM,HGSHELL, 1865
- DYPARAM,HGSOLID, 1866
- DYPARAM,HGTYPE, 1867
- DYPARAM,HICGRAV, 1868
- DYPARAM,HVLFAIL, 1869
- DYPARAM,IMM, 1870
- DYPARAM,INFOBJ, 1871
- DYPARAM,INISTEP, 1872
- DYPARAM,LIMCUB, 1873
- DYPARAM,MATRMERG, 1874
- DYPARAM,MATRMRG1, 1875
- DYPARAM,MAXSTEP, 1876
- DYPARAM,MESHPLN, 1877
- DYPARAM,MINSTEP, 1878
- DYPARAM,MIXGAS, 1879
- DYPARAM,NZEROVEL, 1880

- DYPARAM,OLDLAGT, 1881
- DYPARAM,JWLDET, 1882
- DYPARAM,LIMITER, 1883
- DYPARAM,PARALLEL, 1884
- DYPARAM,PLCOVCUT, 1885
- DYPARAM,PMINFAIL, 1886
- DYPARAM,RBE2INFO, 1887
- DYPARAM,RHOCUT, 1888
- DYPARAM,RJSTIFF, 1889
- DYPARAM,SCALEMAS, 1890
- DYPARAM,SHELLFRM, 1891
- DYPARAM,SHELMYS, 1892
- DYPARAM,SHPLAST, 1893
- DYPARAM,SHSTRDEF, 1894
- DYPARAM,SHTHICK, 1895
- DYPARAM,SLELM, 1896
- DYPARAM,SMP,BATCHSIZ, 1897
- DYPARAM,SMP,CPUINFO, 1899
- DYPARAM,SNDLIM, 1900
- DYPARAM,SPREMAP, 1901
- DYPARAM,SPHERSYM, 1902
- DYPARAM,STRNOUT, 1903
- DYPARAM,TOLCHK, 1904
- DYPARAM,VDAMP, 1905
- DYPARAM,VELMAX, 1906
- DYPARAM,VELMAX1, 1907
- DYPARAM,VISCPLAS, 1909
- ECHOOFF, 1910

- FAILJC, 1964
  - FAILMPS, 1966
  - FAILUDS, 1967
  - FBADLAY, 1969
  - FBALOAD, 1970
  - FBAPHAS, 1971
  - FBODYLD, 1972
  - FBODYSB, 1973
  - FEEDGE, 1974
  - FEFACE, 1977
  - FFCONTR, 1979
  - FFACT, 1981
  - FLOW, 1983
  - FLOWC, 1985
  - FLOWDEF, 1988
  - FLOWT, 1990
  - FLOWUDS, 1993
  - FLSYM, 1995
  - FLUTTER, 1996
  - FORCE, 1998
  - FORCE1, 1999
  - FORCE2, 2000
- ECHOON, 1911
  - EIGB, 1912
  - EIGC, 1914
  - EIGP, 1919
  - EIGR, 1920
  - EIGRL, 1924
  - ELEMUDS, 1928
  - ELEMUDS, 1930
  - ELIST, 1932
  - ENDDATA, 1933
  - ENDDYNA, 1934
  - ENTUDS, 1935
  - EOSDEF, 1937
  - EOSGAM, 1939
  - EOSIG, 1941
  - EOSJWL, 1947
  - EOSMG, 1949
  - EOSNA, 1951
  - EOSPOL, 1953
  - EOSUDS, 1955
  - EPOINT, 1956
  - ERPPNL, 1957
  - EULFOR, 1958
  - EULFOR1, 1959
  - EULFREG, 1960
  - EXCLUDE, 1962
  - EXTRN, 1963

- FTGSEQ , 2060
- GBAG, 2064
- GBAGCOU, 2067
- GENEL, 2068
- GENUDS, 2071
- GMBC, 2072
- GMBNDC, 2080
- GMBNDS, 2082
- GMCONV, 2084
- GMCORD, 2087
- GMCURV, 2089
- GMINTC, 2095
- GMINTS, 2097
- GMLOAD, 2099
- GMNURB, 2102
- GMQVOL, 2105
- GMSPC, 2106
- GMSURF, 2107
- GRAV, 2113
- GRDSET, 2115
- GRID, 2116
- GRIDA, 2118
- GRIDB, 2119
- GRIDF, 2120
- Grids, 2121
- GUST, 2122
- HADACRI, 2123
- HADAPTL, 2126
- HEATLOS, 2128
- HGSUPPR, 2130
- HTRCONV, 2132
- HTRRAD, 2133

■ , 2191

- MACREEP, 2191
- MARCIN, 2193
- MARCOUT, 2195
- MARPRN, 2203
- MASSSET, 2205
- MATBV, 2206
- MAT1, 2207
- MAT1A, 2211
- MAT1F, 2212
- MAT2, 2213
- MAT2F, 2216
- MAT3, 2218
- MAT4, 2220
- MAT5, 2222
- MAT8, 2223
- MAT8A, 2227
- MAT8F, 2231
- MAT9, 2232
- MAT9F, 2234
- MAT10, 2235
- MAT10F, 2237
- MATDEUL, 2238
- MATDIGI, 2240
- MATEP, 2241
- MATEP, 2251
- MATEP, 2266
- MAT1F, 2277
- MATF, 2278
- MATF, 2291
- MATF, 2302
- MATF1, 2308
- MATFAB, 2309
- MATFTG , 2313
- MATG, 2326
- MATHE, 2329
- MATHE, 2337

- MATHED, 2341
- MATHP, 2344
- MATNLE, 2347
- MATORT , 2352
- MATPE1, 2357
- MATRIG, 2359
- MATS1, 2361
- MATS3, 2366
- MATS8, 2368
- MATSMA, 2369
- MATSORT, 2373
- MATT1, 2375
- MATT2, 2377
- MATT3, 2379
- MATT4, 2380
- MATT5, 2381
- MATT8, 2382
- MATT9, 2384
- MATTEP, 2385
- MATTEP, 2387
- MATTF, 2389
- MATTG, 2392
- MATTHE, 2393
- MATTORT, 2396
- MATTUSR, 2397
- MATTVE, 2398
- MATUDS, 2401
- MATUDS, 2404
- MATUSR, 2407
- MATVE, 2410
- MATVE, 2415
- MATVP, 2417
- MAUXCMD, 2422
- MBOLT, 2423
- MBOLTUS, 2425
- MCHSTAT, 2427
- MCOHE, 2429
- MDBCNCT, 2431
- MDBCTB1, 2433
- MDBNDRY, 2435
- MDBOLT, 2436
- MDBULK, 2438
- MDCONCT, 2441
- MDDMIG, 2443
- MDELAM, 2445
- MDEXCLD, 2446
- MDFAST, 2447
- MDLABEL, 2449
- MDLOC, 2450
- MDLPRM, 2452
- MDMIAUX, 2463
- MDMIOUT, 2464
- MDMIR1, 2466
- MDMIR2, 2467
- MDMOVE, 2468
- MDMPC, 2469
- MDMPLN, 2470
- MDRBE2, 2471
- MDRBE3, 2472
- MDRJNT, 2474
- MDROT1, 2475
- MDROT2, 2476
- MDRROD, 2477
- MDSEAM, 2478
- MDTRAN, 2480
- MDWELD, 2481
- MESH, 2486
- MESUPER, 2493
- METADATA, 2496
- MFLUID, 2497
- MGRSPR, 2499
- MINSTAT, 2500
- MISLAND, 2502
- MIXTURE, 2503
- MKAERO1, 2505
- MKAERO2, 2506
- MLAYOUT, 2507
- MNF600, 2508
- MODTRAK, 2510
- MOMAX, 2511
- MOMENT, 2512
- MOMENT1, 2513
- MOMENT2, 2514
- MONCARL, 2515
- MONCNCM, 2517
- MONDSP1, 2518
- MONGRP, 2520
- MONPNT1, 2521
- MONPNT2, 2523
- MONPNT3, 2525
- MONSUM, 2528
- MONSUM1, 2531
- MONSUMT, 2534
- MPC, 2536
- MPCADD, 2537
- MPCAX, 2538
- MPCD, 2539
- MPCREEP, 2540
- MPCY, 2542
- MPHEAT, 2543
- MPROCS, 2544
- MREVERS, 2546
- MRSCON, 2548
- MSTACK, 2550

- MT16SEL, 2551
- MT16SPL, 2553
- MTABRV, 2556
- MTCREEP, 2558
- MTHERM, 2560
- NHRMPRM, 2561
- NLADAPT, 2562
- NLAUTO, 2564
- NLBSH3D, 2569
- NLCYSYM, 2570
- NLDAMP, 2574
- NLFREQ, 2575
- NLFREQ1, 2576
- NLHARM, 2577
- NLHEATC, 2578
- NLMOPTS, 2579
- NLOUT, 2594
- NLOUTUD, 2597
- NLPARM, 2599
- NLPCI, 2607
- NLRGAP, 2609
- NLRSFD, 2613
- NLSTEP, 2616
- NLSTRAT, 2630
- NOLIN1, 2640
- NOLIN2, 2642
- NOLIN3, 2644
- NOLIN4, 2646
- NSM, 2648
- NSM1, 2649
- NSMADD, 2651
- NSMIL, 2652
- NSMIL1, 2653
- NTHICK, 2655
- OMIT, 2656
- OMIT1, 2657
- OMITAX, 2658
- OUTPUT, 2659
- OUTRCV, 2662

- PAABSF, 2665
- PACABS, 2667
- PACBAR, 2668
- PACINF, 2669
- PAERO1, 2670
- PAERO2, 2671
- PAERO3, 2673
- PAERO4, 2675
- PAERO5, 2677
- PANEL, 2681
- PARAM, 2682
- PARAMARC, 2684
- PAXISYM, 2687
- PAXSYMH, 2689
- PBAR, 2691
- PBARL, 2694
- PBARN1, 2700
- PBCOMP, 2704
- PBEAM, 2709
- PBEAM3, 2714
- PBEAMD, 2717
- PBEAML, 2718
- PBELT, 2726
- PBEMN1, 2729
- PBEND, 2733
- PBMARB6, 2741
- PBMNUM6, 2744
- PBMSECT, 2748
- PBRSECT, 2754
- PBSH2DT, 2757
- PBUSH, 2759
- PBUSH1D, 2764
- PBUSH2D, 2769
- PBUSHT, 2771
- PCOHE, 2777
- PCOMP, 2778
- PCOMPA, 2783
- PCOMPF, 2785
- PCOMPFQ, 2787
- PCOMPG, 2788
- PCOMPLS, 2793
- PCONEAX, 2799
- PCONV, 2801
- PCONVM, 2804
- PCONV1, 2806
- PDAMP, 2808
- PDAMP5, 2809
- PDAMPT, 2810
- PDISTB, 2811
- PDISTBM, 2815
- PDUMi, 2817
- PELAS, 2818
- PELAS1, 2819
- PELAST, 2820
- PERMEAB, 2822
- PERMGBG, 2824
- PEULER, 2826
- PEULER1, 2828
- PFAST, 2829
- PFASTT, 2834

- PFTG , 2836
- PGAP, 2840
- PHBDY, 2843
- PINTC, 2844
- PINTS, 2845
- PLCOMP, 2846
- PLOAD, 2850
- PLOAD1, 2852
- PLOAD2, 2855
- PLOAD4, 2857
- PLOADB3, 2860
- PLOADX1, 2862
- PLOTEL, 2865
- PLPLANE, 2866
- PLSOLID, 2868
- PLTSURF, 2869
- PMARKER, 2870
- PMASS, 2871
- PMINC, 2872
- PMREBAI, 2874
- PMREBAR, 2878
- POINT, 2881
- POINTAX, 2882
- PORFCPL, 2883
- PORFBG, 2885
- PORFLOW, 2887
- PORFLWT, 2889
- PORHOLE, 2892

■ , 2956

- QBDY1, 2956
- QBDY2, 2957
- QBDY3, 2958
- QHBDY, 2959
- QSET, 2961
- QSET1, 2962
- QVECT, 2963
- QVOL, 2966
- RADBC, 2968
- RADBND, 2969
- RADC, 2970
- RADCAV, 2972
- RADCOL, 2975
- RADCT, 2976
- RADLST, 2977
- RADM, 2979
- RADMT, 2980
- RADMTX, 2981
- RADSET, 2982
- RANDPS, 2983
- RANDT1, 2985
- RBAR, 2986
- RBAR1, 2988
- RBAX3D, 2990
- RBE1, 2992
- RBE2, 2994
- RBE2GS, 2996
- RBE3, 2999
- RBE3U, 3002
- RBJOINT, 3003
- RCONN, 3006
- RCPARM, 3008
- RCROSS, 3011
- RELEASE, 3013
- RELEX, 3014
- RESTART, 3015
- RFORCE, 3018
- RGYRO, 3022
- RINGAX, 3024
- RINGFL, 3026
- RJONT, 3027
- RLOAD1, 3028
- RLOAD2, 3030
- ROTHYBD, 3032

- ROTOR, 3033
- ROTORAX, 3035
- ROTORG, 3037
- ROTORSE, 3039
- RROD, 3041
- RSPINR, 3043
- RSPINT, 3048
- RSPLINE, 3052
- RSSCON, 3054
- RTRPLT, 3057
- RTRPLT1, 3059
- RVDOF, 3061
- RVDOF1, 3062
- SANGLE, 3063
- SEBNDRY, 3064
- SEBSET, 3065
- SEBSET1, 3066
- SEBULK, 3068
- SECONCT, 3071
- SECSET, 3073
- SECSET1, 3074
- SECTAX, 3076
- SEDLINK, 3077
- SEDRSP2, 3079
- SEDRSP3, 3082
- SEELT, 3085
- SEECLD, 3086
- SELABEL, 3087
- SELOC, 3088
- SEMPLN, 3090
- SENQSET, 3091
- SEQGP, 3092
- SEQSEP, 3093
- SEQSET, 3095
- SEQSET1, 3097
- SESET, 3099
- SESUP, 3100
- SET1, 3101
- SET2, 3102
- SET3, 3103

■ , 3176

- TABD1MD, 3176
- TABDMP1, 3178
- TABL3D, 3181
- TABLE3D, 3186
- TABLED1, 3187
- TABLED2, 3190
- TABLED3, 3192
- TABLED4, 3194
- TABLED5, 3195
- TABLEHT, 3199
- SET4 , 3105
- SETREE, 3106
- SEUSET, 3108
- SEUSET1, 3109
- SHREL, 3110
- SHRPOL, 3111
- SHRUDS, 3112
- SLBDY, 3113
- SLOAD, 3114
- SLOADN1, 3115
- SNORM, 3116
- SPBLND1, 3118
- SPBLND2, 3120
- SPC, 3121
- SPC1, 3123
- SPCADD, 3125
- SPCAX, 3126
- SPCD, 3127
- SPCOFF, 3130
- SPCOFF1, 3131
- SPCR, 3132
- SPHERE, 3134
- SPLINE1, 3135
- SPLINE2, 3137
- SPLINE3, 3139
- SPLINE4, 3141
- SPLINE5, 3143
- SPLINE6, 3145
- SPLINE7, 3148
- SPLINEX, 3150
- SPLINRB, 3152
- SPPOINT, 3153
- SPRELAX, 3154
- STOCHAS, 3155
- SUPAX, 3157
- SUPPORT, 3158
- SUPPORT1, 3159
- SUPPORT6, 3161
- SURFINI, 3163
- SWLDPRM, 3165
- TABLEH1, 3200
- TABLEL1, 3201
- TABLEM1, 3202
- TABLEM2, 3204
- TABLEM3, 3206
- TABLEM4, 3208
- TABLES1, 3209
- TABLEST, 3211
- TABLTG, 3212
- TABLRPC, 3213

- TABLUDS, 3215
- TABRND1, 3216
- TABRNDG, 3218
- TABSCTL, 3219
- TEMP, 3221
- TEMPAX, 3223
- TEMPB3, 3224
- TEMPBC, 3226
- TEMPD, 3229
- TEMPF, 3231
- TEMPN1, 3232
- TEMPP1, 3233
- TEMPP3, 3236
- TEMPRB, 3237
- TERMIN, 3240
- TF, 3242
- THPAD, 3244
- TIC, 3250
- TICEL, 3251
- TICEUL1, 3253
- TICEUDS, 3254
- TICREG, 3255
- TICVAL, 3257
- TIC3, 3259
- TIM2PSD, 3261
- TLOAD1, 3264
- TLOAD2, 3267
- TMPSET, 3271
- TODYNA, 3272
- TOMVAR, 3273
- TOPVAR, 3276
- TRIM, 3280
- TRIM2, 3281
- TRMCPL, 3283
- TSTEP, 3285
- TSTEPNL, 3286
- TTEMP, 3293
- UDNAME , 3294
- UDSESV, 3295
- UNBALNC, 3296
- UNGLUE, 3299
- UNGLUE, 3300
- USET, 3302
- USET1, 3303
- USRSUB6, 3305
- UXVEC, 3306
- VCCT, 3307
- VCCT, 3309
- VIEW, 3312
- VIEW3D, 3314
- VIEWEX, 3316
- WALL, 3321
- WEAR, 3323
- WETELME, 3325
- WETELMG, 3326
- WETLOAD, 3327
- WETSURF, 3329
- YLDHY, 3330
- YLDJC, 3331
- YLDMC, 3333
- YLDMSS, 3334
- YLDPOL, 3336
- YLDRPL, 3337
- YLDSG, 3338
- YLDTM, 3340
- YLDUDS, 3342
- YLDVM, 3344
- YLDZA, 3347

# A

Configuring the Runtime Environment

- Specifying Parameters, 3350
  - Command Initialization and Runtime Configuration Files, 3350
  - Environment Variables, 3354
- User-Defined Keywords, 3355
  - General Keywords, 3355
  - PARAM Keywords, 3356
  - Value Descriptors, 3357
  - Examples:, 3358
- Resolving Duplicate Parameter Specifications, 3360
- Customizing Command Initialization and Runtime Configuration Files, 3363
  - Examples, 3365
- Symbolic Substitution, 3369
  - Introduction, 3369

- Simple Examples, 3369
- Detailed Specifications, 3371
- Examples, 3380

**Na**

Main Index

# Preface

- List of MSC Nastran Books
- Technical Support
- Training and Internet Resources

## List of MSC Nastran Books

A list of some of the MSC Nastran documents is as follows:

|  |
|--|
| <b>Installation and Release Guides</b> |
| ■ Installation and Operations Guide    |
| ■ Release Guide                        |
| <b>Reference Guides</b>                |
| ■ Quick Reference Guide                |
| ■ DMAP Programmer's Guide              |
| ■ Reference Guide                      |
| ■ Utilities Guide                      |
| <b>Demonstration Guides</b>            |
| ■ Linear Analysis                      |
| ■ Implicit Nonlinear (SOL 400)         |
| ■ Explicit Nonlinear (SOL 700)         |
| <b>User's Guides</b>                   |
| ■ Getting Started                      |
| ■ Linear Static Analysis               |
| ■ Dynamic Analysis                     |
| ■ Embedded Fatigue                     |
| ■ Embedded Vibration Fatigue           |
| ■ Thermal Analysis                     |
| ■ Superelements and Modules            |
| ■ Design Sensitivity and Optimization  |
| ■ Rotordynamics                        |
| ■ Implicit Nonlinear (SOL 400)         |
| ■ Explicit Nonlinear (SOL 700)         |
| ■ Aeroelastic Analysis                 |
| ■ User Defined Services                |
| ■ Non Linear (SOL 600)                 |
| ■ High Performance Computing           |
| ■ DEMATD                               |

You may find any of these documents from MSC Software at:

<http://simcompanion.mscsoftware.com/infocenter/index?page=home>

## Technical Support

For technical support phone numbers and contact information, please visit:

<http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx>

**Support Center** (<http://simcompanion.mscsoftware.com>)

The SimCompanion link above gives you access to the wealth of resources for MSC Software products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from MSC Software, allows you to manage your support cases, and participate in our discussion forums.

## Training and Internet Resources

### MSC Software ([www.mssoftware.com](http://www.mssoftware.com))

MSC Software corporate site with information on the latest events, products and services for the CAD/CAE/CAM marketplace.

<http://simcompanion.mssoftware.com>

The SimCompanion link above gives you access to the wealth of resources for MSC Software products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from MSC Software, allows you to manage your support cases, and participate in our discussion forums.

<http://www.mssoftware.com/msc-training>

The MSC-Training link above will point you to schedule and description of MSC Seminars. Following courses are recommended for beginning Nastran users.

#### NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran

This course serves as an introduction to finite element analysis. It includes discussion of basic features available in MSC Nastran for solving structural engineering problems. In this course, all finite element models will be created and edited using a text editor, not a graphical pre-processor. Proper data structure of the MSC Nastran input file is covered. At the conclusion of seminar, the student will be familiar with fundamental usage of MSC Nastran.

#### NAS101B - Advanced Linear Analysis using MSC Nastran

This course is a continuation of NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran. In this course, you will learn:

- Theory of buckling analysis and how to perform a buckling analysis,
- About rigid elements - MPC, RBAR, RBE2 and RBE3,
- Modeling with interface element, CINTC and connectors,
- Lamination theory and composite materials,
- MSC Nastran composite theory, Failure theories, Linear contact and permanent glued contact
- Different model checks Modeling tips and tricks.

#### NAS120 - Linear Static Analysis using MSC Nastran and Patran

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC Nastran and Patran. MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran will be an integral part of the examples and workshops and will be used to generate and verify illustrative MSC Nastran models, manage analysis submission requests, and visualize results. This seminar provides the foundation required for intermediate and advanced MSC Nastran applications.

# 1

## Executing MSC Nastran

- Executing MSC Nastran

## Executing MSC Nastran

MSC Nastran is executed from the command line using the command `nast20200`. This command sets environment variables in order for MSC Nastran to correctly execute on your system and it also has capabilities to predict memory and solvers (as of MSC Nastran 2020 with `solve=auto`). The basic format of the `nast20200` command is

```
nast20200 input_data_file keywords
nast20200 input_data_file [keyword1=value1 keyword2=value2 ...]
```

where `input_data_file` is the name of the file containing the input data and `keyword=valuei` is one or more optional keyword assignment arguments. For example, to run an MSC Nastran job using the data file `example1.dat`, enter the following command:

```
nast20200 example1
```

Most keyword assignments can be specified as command line arguments and/or included in RC files. There are some exceptions which may be specified on the command line or the User RC files, but not in the system RC files (`MSC_BASE/conf/RCfile`).

There are two RC files controlled by the user:

- The user RC file is in your home (or login) directory. This file should be used to define parameters that are applied to all jobs run by the user.
  - The local RC file is `nast20200rc` on Linux and `NAST20200.rcf` on Windows, and is located in the same directory as the input data file. If the “`rcf`” keyword is used, this local RC file is ignored. This file should be used to define parameters that are applied to all jobs contained in the input data file directory.
1. The tilde (~) character is not recognized within RC files.
  2. Environment variables are only recognized when used in the context of a logical symbol (on Windows `NAST20200.rcf`) (see [Using Filenames and Logical Symbols](#) in the *MSC Nastran Installation and Operations Guide*).
  3. When a keyword is specified on the command line, embedded spaces or special characters that are significant to the shell must be properly quoted; quotes should not be used within RC files.

The keywords listed as follows are the most common for various computers, but are not available on all computers. Also, the defaults may be site-dependent. Please consult your [Keywords and Environment Variables](#) in the *MSC Nastran Installation and Operations Guide* for keyword applicability, specialized keywords, and further discussion of the keywords and their defaults. Keywords that use yes/no values accept partial specification and case-independent values. For example, “yes” may be specified as “y”, “ye”, or “yes” using uppercase or lowercase letters. The examples assume the jobs are run under a Linux operating system.

|         |  |                      |
|---------|--|----------------------|
| after   | <code>after = time</code>  | Default: <i>None</i> |
| (Linux) | Holds the job’s execution until the time specified by <code>time</code> . See the description of the “at” command in your system documentation for the format of <code>time</code> |                      |

Example:                    `nast20200 example after=10:00`

The job is held until 10:00 AM.

|   |  |                                 |
|---|--|---------------------------------|
| append  | append={yes no}  | Default: no                     |
| Combines the .f04, .f06, and .log files into a single file after the run completes. If "no" is specified, the files are not combined. If "yes" is specified, the files are combined into one file with the suffix ".out".   |  |                                 |
| Example:  | nast20200 example append=yes   |                                 |
| Result: The .f04, .f06, and .log files are combined into a file named example.out.  |  |                                 |
| batch   | batch={yes no}   | Default: yes                    |
| (Linux)   | Indicates how the job is to be run. If "yes" is specified, the job is run as a background process. If "no" is specified, the job is run in the foreground. If the "aft" or "queue" keywords are specified, the batch keyword is ignored. Jobs submitted with "batch=yes" will run under nice(1). |                                 |
| <b>Note:</b> If the job is already running in an NQS or NQE batch job, the default is "no".   |  |                                 |
| bpool   | bpool= <i>value</i>  | Default: <i>See text below.</i> |
| Specifies the number of GINO and/or executive blocks, or memory size in MB, GB, etc., that are placed in buffer pool.   |  |                                 |
| The size is specified as the number of blocks (BUFFSIZE words long), a percentage of MEM, or the number of words or bytes followed by one of the modifiers: "T", "TW", "TB", "G", "GW", "GB", "M", "MW", "MB", "K", "KW", "KB", "W", "B". See Specifying Memory Sizes, 63 for a description of these modifiers. |  |                                 |
| If mem=max (which is the default) or/and solve=auto is not used, the default of bpool is 150 GINO blocks.   |  |                                 |
| If solve=auto is used, bpool will be set automatically.   |  |                                 |
| If mem=max is used, the default of bpool will be set to:  |  |                                 |
| <ul style="list-style-type: none"> <li>■ 25% of memory for non SOL 101 or SOL 400.</li> <li>■ the remaining memory after memory estimate needed for the solver for SOL 101 or SOL 400.</li> </ul>   |  |                                 |
| buffsize  | buffsize= <i>value</i>   | Default: 32769 words            |
| Use to control the number of words per I/O record.  |  |                                 |

For large models, it is recommended to change bufsize to 65537, which is the maximum allowed.

casi If set to "no", this flag will disable the Casi solver as a possible option when "solve=auto" is specified.

**dbs**                    **dbs=***pathname*                    Default= . {Current directory}

Creates database files (see [Using the NASTRAN Statement](#) in the *MSC Nastran Installation and Operations Guide*) using an alternate file prefix. If “dbs” is not specified, database files are created in the current directory using the basename of the input data file as the prefix. If the “dbs” value is a directory, database files are created in the specified directory using the basename of the input data file as the filename.

**Note:** If “dbs” is specified and “scratch=yes” is specified, a warning will be issued and “scratch=no” is assumed.

In the following examples, assume the current directory includes subdirectories “mydir” and “other”, and that an “example.dat” exists in both the current directory and “other”. That is, ./example.dat, ./mydir/, ./other, and ./other/example.dat exist on Linux, and .\example.dat, .\mydir\, .\other, and .\other\example.dat exist on Windows.

Example: nast20200 example

Result: Database files are created in the current directory with the name “example” e.g., ./example.DBALL on Linux; and .\example.DBALL on Windows.

Example: nast20200 other/example

Result: Database files are created in the “other” directory with the name “example”, e.g., `../other/example.DBALL` on Linux, and `\other\example.DBALL` on Windows.

Result: Database files are created in the current directory with the name "myfile", e.g., ./myfile.DBALL on Linux, and .\myfile.DBALL on Windows.

Result: Database files are created in the mydir directory with the name "example", e.g., ./mydir/example.DBALL on Linux, and .\mydir\example.DBALL on Windows.

Result: Database files are created in the mydir directory with the name "myfile", e.g., ./mydir/myfile.DBALL on Linux, and .\mydir\myfile.DBALL on Windows

Example: nast20200 example dmp=4 host=a:b:c:d  
dbs=/aa:/bb:/cc:/dd

Result: This example will set the “dbs” directory to “/aa” on host a, “/bb” on host b, “/cc” on host c, and finally “/dd” on host d.

**Note:** The use of distinct per-task database directories can have a significant impact on elapsed time performance of DMP jobs on SMP and NUMA systems.

|     |                          |  |
|-----|--------------------------|--|
| dmp | dmp= <i>dmp parallel</i> | Default: No parallelism based upon dividing the model  |
|     |                          | The dmp keyword specifies the number of processors used to efficiently obtain a solution to the simulation. These processors may be on the same node or on multiple nodes specified by the HOST command line option.   |
|     |                          | Specifying dmpparallel > 1 and not including a DOMAINSOLVER Executive Command Statement has the following effects.   |
|     |                          | <ul style="list-style-type: none"><li>■ The extraction of real eigenvalues using the default Lanczos method will be performed in parallel in SOL 103 and 111.</li><li>■ The frequency response using SOL 111 will be performed in parallel.</li></ul>  |
|     |                          | Specifying dmp > 1 and using the DOMAINSOLVER provides the following addition user controlled performance gains using parallel processing  |
|     |                          | <ul style="list-style-type: none"><li>■ The extraction of eigenvalues may be performed using Automated Component Modal Synthesis (ACMS) which is an approximate method that is very computational efficient when a large number of modes (high frequencies) are required.</li><li>■ Design sensitivity may be performed in parallel in SOL 200</li><li>■ The formation of using advanced nonlinear elements stiffness matrices may be performed in parallel for SOL 400.</li></ul> |
|     |                          | It may also be necessary to define a host file if a network based distributed computing is used. See the <i>MSC Nastran Installation and Operators Guide</i> .   |

**gpuid**                    gpuid=id,id or gpuid=id:id                    Default: none  
 id: the ID of a licensed GPU device to be used in the analysis.

In MSC Nastran 2019 Feature Pack 1 release, the GPUs are used for two types of operations: matrix factorization and matrix multiplication. In the routines that use GPUs for matrix factorization, only one GPU will be used per DMP process. So in order to use multiple GPUs, the user must also use multiple DMP processes:

*prod\_ver* job gpuid=0,1,...,ngpu-1 dmp=ndmp, where ndmp≥ngpu.

Each DMP process will be assigned a GPU ID in round robin fashion.

In MPYAD and FASTFR modules, on the other hand, multiple GPUs can be used for matrix multiplication by a single process, as long as the number of SMP-threads is equal to or larger than the number of GPUs. These modules also use multiple parallel streams for data transfer between the GPU and the host system, so it is recommended to use the maximum number of available CPU cores/threads because that helps reduce the overhead of data transfer between the host and the GPUs.

So, for example, in order to achieve the best performance on a system with 20 CPU cores and 2 GPUs the user is advised to run Nastran with:

*prod\_ver* job dmp=2 smp=10 gpuid=0,1

or

*prod\_ver* job dmp=4 smp=5 gpuid=0,1.

Only up to 8 CUDA-compatible GPUs can be supported.

**gpu\_min\_rank**                    gpu\_min\_rank=value                    Default: 32

The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via **gpu\_min\_front** and its companion parameter, **gpu\_min\_rank**. The value specified must be an integer greater or equal to 1. If the rank of the frontal matrix is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. This keyword may also be set via SYSTEM cell 655.

gpu\_min\_front      gpu\_min\_front=value      Default: 2048

The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via gpu\_min\_front and its companion parameter, gpu\_min\_rank. The value specified must be an integer greater or equal to 1. If the front size is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. This keyword may also be set via SYSTEM cell 656.

memorymax      memorymax=*maximum user memory*      Default: 0.5 x the amount of physical memory on your machine

The user can enter the amount of memory as *n*GB or as *f*xphysical; *f* is the fraction between 0 and 1.

When dmp is used, the amount of memory per processor is memorymax / dmp.

If the machine is going to be used by a single user or a single job, it is recommended to set memorymax to 0.75xPhysical.

If multiple Nastran jobs are running simultaneously on the same node of a computer system, it is recommended to set memorymax to 0.8\*physical\_memory divided by the number of possible Nastran jobs running on the same node. Otherwise, one might encounter an error due to lack of memory on small models. The error can be avoided by using above setting in the system RC file or mem=size specified properly for the job in the command line.

memory      memory=*memory\_size*      Default=max

Specifies the amount of memory to allocate. The *memory\_size* can be specified either as a number of words, or as a number followed by one of the following modifiers:

G or Gw      Multiply *memory\_size* by 1024\*\*3.

Gb      Multiply *memory\_size* by (1024\*\*3)/*bytes\_per\_word*.

where *bytes\_per\_word* is 8 or you can enter a fraction of the physical memory

If "memory=estimate" is specified, ESTIMATE will be used to determine size. If "memory=max" is specified, then the amount of memory allocated depends on the "memorymaximum" keyword.

Example:      nast20200 example memory=5gb

Result: The job is run using a memory size of 5 gigabytes.

Example:      nast20200 example  
memory=0.5xPhysical

Result: The job is run using one-half the amount of physical RAM.

For a more detailed description, see the [memory](#) (Ch. B) in the *MSC Nastran Installation and Operations Guide*.

news news={yes|no|auto} Default=yes  
Displays the news file (*install\_dir*\msc20200\nast\news.txt on Linux and *install\_dir*\msc20200\nast\news.txt on Windows) in the .f06 file. If “auto” is specified, the news file is only displayed if it has been modified since the last time it was displayed for you. If “yes” is specified, the news file is displayed in the .f06 file regardless of when it was last changed. If “no” is specified, the news file is not displayed in the .f06 file.

Example: nast20200 example news=yes

Result: The news file is displayed in the .f06 file after the title page block.

notify notify={yes|no} Default=yes  
Sends notification when the job is completed. See the “ncmd” keyword to define an alternate notification command.

**Note:** If the job is queued using the queue keyword, or the job is already running in an NQS batch job, the default is “no”.

Example: nast20200 example notify=yes

old old={yes|no} Default=yes  
Saves previous copies of the .f04, .f06, .log, .op2, .out, pch, and .plt output files using sequence numbers (additional user-specified file types can be versioned with the “oldtypes” keyword). Sequence numbers are appended to the keyword filenames and are separated by a period.

If “yes” is specified, the highest sequence number of each of the output files is determined. The highest sequence number found is incremented by one to become the new sequence number. Then, all current output files that do not include sequence numbers are renamed using the new sequence number as a type.

Example: nast20200 example old=yes

For example, assume your current working directory contains the following files:

v2401.dat v2401.f04.1 v2401.f06 v2401.log v2401.log.1  
v2401.f04 v2401.f04.2 v2401.f06.1 v2401.log.1 v2401.log.3

Apparently, the user ran the job four times, but deleted some of the files; e.g.

v2401.f04.3

v2401.f06.3

When the job is run again with “old=yes”, the files are renamed as follows:

v2401.f04 is renamed to v2401.f04.4

v2401.f06 is renamed to v2401.f06.4

v2401.log is renamed to v2401.log.4.

The sequence number 4 is used because it is one greater than the highest sequence number of all of the selected files (the highest being v2401.log.3).

out                    *out=pathname*                    Default= .  
Saves the output files using a different file prefix or in a different directory. If “out” is not specified, the output files are saved in the current directory using the basename of the input data file as a prefix. If the “out” value is a directory, output files are created in the specified directory using the basename of the input data file as the filename.

In the following examples, assume the current directory includes subdirectories “mydir” and “other”, and that an “example.dat” exists in both the current directory and “other”. That is, ./example.dat, ./mydir, ./other, and ./other/example.dat exist on Linux; and .\example.dat, .\mydir, .\other, and .\other\example.dat exist on Windows.

Example:                    nast20200 example  
or:                            nast20200 other/example

Result: Output files are created in the current directory with the name “example”, e.g., ./example.f06 on Linux and .\example.f06 on Windows.

Example:                    nast20200 example out=myfile

Result: Output files are created in the current directory with the name “myfile”, e.g., ./myfile.f06 on Linux and .\myfile.f06 on Windows.

Example:                    nast20200 example out=mydir

Result: Output files are created in the mydir directory with the name “example”, e.g., ./mydir/example.f06 on Linux and .\mydir\example.f06 on Windows.

Example:                    nast20200 example out=mydir/myfile

Result: Output files are created in the mydir directory with the name “myfile”, e.g., ./mydir/myfile.f06 on Linux and .\mydir\myfile.f06 on Windows.

**rcf**      rcf=pathname                  Default=no  
Specifies the name of the local RC file. If this keyword is not specified, the nast20200rc file on Linux and NAST20200.rcf on Windows located in the input data file's directory is used

Example:                   \$ nast20200 example rcf=nast.rc

Result: The nastran command will process ./nast.rcf on Linux, or .\nast.rcf on Windows in lieu of the default local RC file ./nast20200rc on Linux, and .\nAST20200.rcf on Windows.

scratch scratch={yes|no|mini|post} Default=no

Deletes the database files at the end of the run. If the database files are not required, “scratch=yes” can be used to remove them, thereby preventing cluttering of the directory with unwanted files. If “mini” is specified, a reduced size database (that can only be used for data recovery restarts) will be created. See [Database Concepts](#) in the *MSC Nastran Reference Guide* for further details on the “mini” database. If scratch=post is specified, a reduced size database intended for use by Patran or the toolkit will be created. Scratch=post also performs the actions of NASTRAN INDEX=19. But with scratch=post the resulting database is not restartable.

All database files created by the run are deleted at the end of the job in the same way as the FMS statement INIT MASTER(S).

`sdirectory`      `sdirectory=directory`      Default: *See the description below.*

See [Determining Resource Requirements](#) in the *MSC Nastran Installation and Operations Guide* for information on estimating a job's total disk space requirements.

Specifies the directory to use for temporary scratch files created during the run. MSC Nastran can create very large scratch files; therefore, the scratch directory should contain sufficient space to store any scratch files created during a run. You must have read, write, and execute privileges to the directory.

Linux: The default value is taken from the TMPDIR environment variable if it is set to a nonnull value. Otherwise, the computer's default temporary file directory is chosen; this is usually /tmp.

Windows: The default value is taken from the TEMP environment variable.

Linux Example:                    nast20200 example sdir=/scratch

Result: Scratch files are created in the directory /scratch.

|  |                                     |                      |
|--|-------------------------------------|----------------------|
| smp  | smp= <i>value</i>                   | Default = 0          |
| Specifies the maximum number of processors selected for shared-memory parallel (SMP) processing in several numeric modules. In some cases SMP parallelization can be combined with DMP. In such cases, the smp keyword specifies the number of threads per process used in the simulation.   |                                     |                      |
| smp parallelization is available for mathematical kernels.   |                                     |                      |
| smp parallelization is available for the Intel MKL PARDISO solver in SOL 101, 107, 108, 111, 200, and 400; see the SPARSE SOLVER section of this document.   |                                     |                      |
| smp parallelization is available for the NLEMG modules and is disabled in the following cases:   |                                     |                      |
| <ul style="list-style-type: none"> <li>■ heat transfer and coupled analysis simulations;</li> <li>■ models that have contact between Advanced elements and Nastran elements;</li> <li>■ models that have QUADR/QUAD4 or TRIAR/TRIA3 element types arranged into the same element groups.</li> <li>■ models that have friction.</li> </ul>  |                                     |                      |
| If models have contact between Nastran and Advanced elements resulting in a SYSTEM FATAL MESSAGE (SFM) with smp > 1, then rerun the job with sys107=65536+ <i>number</i> . For example, if smp=8 is desired but an SFM results, then rerun with sys107=65544 in replace of smp=8. This turns off the SMP parallelization for NLEMG.  |                                     |                      |
| solve  | solve={auto train}                  | Default (not set)    |
| If solve=auto then Nastran will automatically select the solver/parallel/memory. If solve=train then Pardiso memory coefficients will be updated. These options may be placed on the command line or in the User's RC files.   |                                     |                      |
| symbol   | symbol= <i>name</i> = <i>string</i> | Default: <i>None</i> |
| Defines a symbolic (or logical) name used in ASSIGN and INCLUDE statements and in command line arguments. This keyword may be specified in initialization or RC files and on the command line. The symbol definition may include references to previously defined symbols or environment variables using the standard "\$ <i>name</i> " or "\${ <i>name</i> }" syntax on Linux or % <i>name</i> % syntax on Windows. For convenience, the character separating the "symbol" and "name" specification and the "name" and "string" specification may be either an equal sign ("=") or a hash mark ("#"). The use of a hash mark allows this keyword to be specified as an argument to a Windows .bat file. |                                     |                      |
| If "node" is specified, symbolic names defined using this keyword are not used on the local system. Instead the specified values are passed to the remote system. This means that any pathnames must be valid on the remote system. Use the "lsymbol" keyword to specify symbolic names for the local system.  |                                     |                      |
| If "node" is not specified, symbolic names defined using the "lsymbol" keyword are processed as if they were defined using the "symbol" keyword.   |                                     |                      |

Symbolic names are processed in the order they are encountered while processing the initialization and RC files and the command line. If a duplicate symbolic name is encountered, the new value replaces the previously specified value.

Symbolic names must be 16 characters or less. The value assigned to the symbolic name must be 256 characters or less. If the symbolic name used in an ASSIGN or INCLUDE statement or in command line arguments is not defined, it is left in the filename specification as is.

For example, many of the .tpl and .demo input data files have ASSIGN statements such as the following:

```
ASSIGN 'MASTER=DBSDIR:abc.master'
```

The string "DBSDIR:" specifies a symbolic name that is to be replaced by another string. The replaced string is defined by the "symbol=" keyword (or "lsymbol=" keyword if "node" was not specified) in an initialization or RC file, on the command line, or as environment variable. For example,

(Linux)

```
symbol=DBSDIR=/dbs
```

(Windows)

```
symbol=DBSDIR=d:\dbs
```

When the previous ASSIGN statement is processed, the filename assigned to the logical name MASTER is /dbs/abc.master on Linux and d:\dbs\abc.master on Windows. An alternate way of defining symbolic names is through the use of environment variables. For example, typing the following command

```
export DBSDIR=/dbs
```

at a Korn shell prompt, or

```
setenv DBSDIR /dbs
```

at a C-shell prompt, or

at a Windows shell prompt, is equivalent to the "symbol" keyword definition.

**Note:** If a symbolic name is defined by both a symbol statement in an RC file and by an environment variable, the symbol statement value will be used.

The section titled [Environment Variables](#) in the *MSC Nastran Installation and Operations Guide* contains a list of environment variables that are automatically created by the nastran command. Of particular interest to the logical symbol feature are the OUTDIR and DBSDIR variables. These variables refer to the directory that will contain the output files (set using the "out" keyword) and the directory that will contain the permanent database files (set using the "dbs" keyword), respectively.

|   |                             |                           |
|---|-----------------------------|---------------------------|
| uds   | uds={filename   model}      | Default: None             |
| The file name is the source file that defines user subroutines. This keyword is used to build user service at nastran job submittal time. The user subroutines implementations are from the specified file and the user service name is from the connect service statement in input file. |                             |                           |
| On windows the UDS files being created may not be in "C:\Program Files". Please copy files to a writable directory to build.  |                             |                           |
| Example:  |                             |                           |
| If user routine is written in Fortran: <code>nast20200 example uds=mysource.F</code>  |                             |                           |
| If user routine is written in C++: <code>nast20200 example uds=mysource.cpp</code>  |                             |                           |
| The mysource.F includes all user defined subroutines. There may be multiple user subroutines from multiple interfaces in the file. The user subroutine names are predefined and documented in User Defined Services.  |                             |                           |
| Another option to define user subroutines is to put them in BEGIN BULK UDS section in the input file. The content in this section will be used to create source file when building user service. If this option is used, the uds keyword should be set to model.                          |                             |                           |
| Example: <code>nast20200 example uds=model</code>   |                             |                           |
| udssave   | udssave=pathname            | Default: output directory |
| The udssave is the location to build user service. If udssave is given, the path will be used to build and save user service. If it is not given, the nastran output directory will be used and the user service will be removed after nastran job run.                                   |                             |                           |
| Example: <code>nast20200 example uds=mysource.F udssave=/scratch/mydir</code>   |                             |                           |
| xdbold  | xdbold={yes no} Default=yes |                           |
| Keeps or deletes the previous copy of the .xdb.   |                             |                           |
| If "yes" is specified, the previous copy is kept and appended by the current run.   |                             |                           |
| If "no" is specified, the previous copy is deleted and a new file is created. This gives the same behavior as below FMS statement:  |                             |                           |
| <code>ASSIGN dbc=&lt;file_name&gt;.xdb delete</code>  |                             |                           |
| Example: <code>nast20200 example xdbold=no</code>   |                             |                           |



# 2

# NASTRAN Statement

- The NASTRAN Statement

## The NASTRAN Statement

The NASTRAN statement is used to specify values for certain Executive System operational parameters. These parameters are also called system cells. The NASTRAN statement is used for exceptional circumstances and is therefore not needed in most runs. The NASTRAN statement may also be specified in the runtime configuration (RC) files at the system, user, and job level as described in the [MSC Nastran 2020 Installation and Operations Guide](#).

## NASTRAN

## Executive System Parameter Modification

Specifies values for certain Executive System operational parameters called system cells.

**Format:**

NASTRAN cellnamei=expressioni, ..., cellnamen=expressionn

or

NASTRAN SYSTEM(i)=expressioni, ..., SYSTEM(n)=expressionn

| Descriptor | Meaning  |
|------------|--|
| cellnamei  | System cell names from <a href="#">Table 2-1</a> .   |
| SYSTEM     | Specifies the system cell number.  |
| expression | See DEFINE statement for description.  |
| i          | System cell number from <a href="#">Table 2-1</a> or from the SYSTEM common block described in the <i>MSC Nastran User Modifiable User's Guide</i> . |

**Remarks:**

1. The NASTRAN statements may appear anywhere in the File Management Section. The NASTRAN statement may also be specified in runtime configuration (RC) files. See [Customizing Command Initialization and Runtime Configuration Files](#) (App. A) in the .
2. System cell values and their associated cell names may also be set with the DEFINE statement. They may also be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See [PUTSYS, GETSYS](#) in the *DMAP Programmer's Guide*.
3. More than one NASTRAN statement and/or DEFINE statement may be present and, if a system cell is specified more than once among these statements, then the last specification takes precedence.
4. The expression will use type conversion rules based on the type (i.e., integer, real, or logical) of the cellname, as defined on a previous DEFINE statement (see the DEFINE statement for conversion rules).
5. If expression is omitted, the system cell associated with the cellname will be assigned the value as set on a previous DEFINE statement.

**Examples:**

1. Either of the following statements could be used to change the default value for block size.

NASTRAN SYSTEM (1) = 4097

or

NASTRAN BUFFSIZE = 4097

or, if a prior DEFINE statement had defined a keyword MY\_SYSBUF to the value 4097, then the following code could be used:

NASTRAN SYSTEM(1)=MY\_SYSBUF

or

NASTRAN BUFFSIZE=MY\_SYSBUF

The following statement is used to request execution of MSGMESH:

NASTRAN MESH

2. Table 2-1 gives a summary of the recommended system cells. System Cells

Table 2-1      System Cell Summary

| System Cell Name (Number) | Function and Reference   |                                |
|---------------------------|--|--------------------------------|
| BUFFSIZE (1)              | Specifies the number of words in a physical record. Also called block length.  |                                |
| F06 (2)                   | Specifies FORTRAN unit number for standard output file (.f06). (Integer $\geq 0$ ; a value of 0 sends the results to the log file; Default=6). This system cell may not be set for SOL 600 or SOL 700.   |                                |
| NLINES (9)                | Specifies the number of lines printed per page of output. <a href="#">LINE (Case), 436</a> .   |                                |
| MAXLINES (14)             | <a href="#">MAXLINES (Case), 446</a>   |                                |
| METIME (20)               | Minimum time for execution summary table message. <a href="#">Output Description</a> in the <i>MSC Nastran Reference Guide</i> .   |                                |
| APP (21)                  | Approach Flag. See the <a href="#">APP, 115</a> Executive Control statement. If APP HEAT is specified, then this system cell is set to 1.  |                                |
| MACHTYPE (22)             | Machine type   |                                |
| DIAGA (25)                | Alternate method to set DIAGs 1 through 32. <a href="#">DIAG, 122</a> .  |                                |
| CONFIG (28)               | Machine subtype  |                                |
| MESH (31)                 | Requests execution of MSGMESH.   |                                |
| ADUMi (46 - 54)           | Dummy element flag, i=1 through 9.   |                                |
| HEAT (56)                 | <a href="#">APP, 115</a>   |                                |
|                           | 0  | Structural analysis (Default). |
|                           | 1  | Heat transfer                  |
| DIAGB (61)                | Alternate method to set diagnostics 33 through 64. <a href="#">DIAG, 122</a> .   |                                |
| PUNCH (64)                | Specifies FORTRAN unit number for PUNCH file (.f07). (Default=7)   |                                |
| MPYAD (66)                | <a href="#">MPYAD</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> .  |                                |
| DCMPOUT(69)               | Controls matrix decomposition for MSCLDL and MSCLU. Same as <a href="#">DECOMP</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> and the <a href="#">Option Selection</a> (p. 77) in the <i>MSC Nastran Numerical Methods User's Guide</i> . |                                |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number)     | Function and Reference  |  |
|-------------------------------|---|--|
|                               | 0 or -1   | Print up to 50 messages for null columns and zero diagonals (Default=-1).  |
|                               | 1   | Terminates execution when first null column is encountered.  |
|                               | 2   | Suppress printing of message when a null column is encountered.  |
|                               | 4   | Terminates execution when first zero diagonal term is encountered.   |
|                               | 16  | Place 1.0 in diagonal position for all null columns and proceed with the decomposition.  |
|                               | 32  | Terminates execution on zero diagonal term.  |
|                               | 64  | Exit after execution of preface for symmetric decomposition.   |
|                               | 256   | Print MAXRATIO messages (SIM 4159) from DCMP.  |
| <a href="#">DELF (77)</a>     | Deletes form feeds.   |  |
| <a href="#">DBSET</a>         | Database neutral file set. <a href="#">SubDMAP DBFETCH in the MSC Nastran DMAP Programmer's Guide</a> .   |  |
| <a href="#">DMAP (82)</a>     | Allows NOGO to operate. See Processing of User Errors <i>in the MSC Nastran DMAP Programmer's Guide</i> .   |  |
| <a href="#">F04 (86)</a>      | Specifies FORTRAN unit number for Execution Summary Table (.f04). <a href="#">Output Description in the MSC Nastran Reference Guide</a> (Integer $\geq 0$ ; a value of 0 sends the results to the log file; Default = 4). |  |
| <a href="#">RADMTX (87)</a>   | Type of radiation exchange coefficients, <a href="#">RADMTX</a> .   |  |
|                               | 1   | Direct input of a symmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the symmetry, only one-half of the RADMTX may be entered.   |
|                               | 2   | Direct input of an unsymmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the unsymmetry, the full matrix must be specified on the RADMTX entries.   |
|                               | 3   | If you are running a view factor calculation in an opened enclosure, NASTRAN assumes that the radiation will be lost to space at absolute zero degrees Kelvin. You can set SYSTEM(87)=3 so that radiation will not be lost to space. |
| <a href="#">RADLST (88)</a>   | Print radiation area summary. <a href="#">RADLST, 2977</a> .  |  |
| <a href="#">SMP (107)</a>     | Number of processors used for Shared Memory Parallel  |  |
| <a href="#">NEWHESS (108)</a> | Request complex eigenvalue method. See the <a href="#">EIGC, 1914</a> entry, <i>MSC Nastran Numerical Methods User's Guide</i> .  |  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
| (109)                     | Controls DMAP execution:   |   |
|                           | 0  | Do not execute DMAP instruction if all outputs are previously computed. |
|                           | 1  | Always execute DMAP instruction (Default).                              |
| BUFFPOOL (114)            | Bufferpool size. <a href="#">Keywords and Environment Variables</a> in the <i>MSC Nastran Installation and Operations Guide</i> .  |   |
| (119)                     | Controls the type of bufferpool method to be used:   |   |
|                           | 4  | Use Version 2012.2 bufferpool method. Default = 4.                      |
|                           | Not 4  | Use Version 2012.1 bufferpool method.                                   |
| ATTDEL (124)              | Controls the automatic assignment of the delivery database. <a href="#">Database Concepts</a> in the <i>MSC Nastran Reference Guide</i> . See also <a href="#">Creating and Attaching Alternate Delivery Databases</a> in the <i>MSC Nastran Installation and Operations Guide</i> . |   |
|                           | 0  | Enables automatic assigning (Default).                                  |
|                           | -1   | Disables automatic assigning.   |
| NOKEEP (125)              | Controls NOKEEP option of the RESTART File Management statement.   |   |
|                           | 0  | Disable NOKEEP  |
|                           | 1  | Enable NOKEEP   |
| SPARSE (126)              | Sparse matrix method selection for MSCLDL and MSCLU. For unsymmetric sparse matrix decomposition method selection, see cell number 209. The following values may be summed in order to select sparse matrix methods in the operations listed below:                                  |   |
|                           | 0  | Deactivate sparse methods   |
|                           | 1  | Multiplication  |
|                           | 8  | Symmetric decomposition   |
|                           | 16   | Forward-backward substitution   |
|                           | The default is 25, which is the sum of all values.   |   |
| UPDTIM (128)              | Specifies database directory update time.<br><a href="#">DBUPDATE, 90</a> FMS statement.   |   |
|                           | 0  | Do not update.  |
|                           | >0   | Time, in minutes, between database directory updates.                   |
| SMPYAD67 (129)            | Select pre-Version 67 method in the SMPYAD module. <a href="#">SMPYAD</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> .  |   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |   |
|---------------------------|---|---|
|                           | 0   | Use current method (Default).                         |
|                           | 1   | Use pre-Version 67 method.                            |
| <b>MAXDBSET</b>           | The maximum number of online DBsets attached to the run.  |   |
| <b>AUTOASGN (133)</b>     | Controls autoassigning of dbsets. Sum the desired values.<br>(Default=7). <a href="#">Database Concepts in the MSC Nastran Reference Guide</a> .  |   |
|                           | 0   | No databases are automatically assigned.              |
|                           | 1   | Only the primary database is automatically assigned.  |
|                           | 2   | Only the delivery database is automatically assigned. |
|                           | 4   | Only located databases are automatically assigned.    |
| <b>TSTAMP (135)</b>       | Controls timestamp checking of DBsets.  |   |
|                           | 0   | Do not check  |
|                           | 1   | Check (Default)                                       |
|                           | 2   | Same as 1 and print diagnostics                       |
| <b>QUADINT (141)</b>      | Specifies quadratic or linear interpolation for the line search method in nonlinear analysis (SOL 106, SOL 129, and SOL 153).   |   |
|                           | 0   | Quadratic interpolation (Default)                     |
|                           | 1   | Linear interpolation                                  |
| <b>SCR300 (142)</b>       | Requests creation of SCR300 partition on SCRATCH DBset.<br><a href="#">INIT</a> , <a href="#">98</a> FMS statement.   |   |
|                           | 1   | Do not create SCR300 partition.                       |
|                           | 2   | Create SCR300 partition (Default).                    |
| <b>LOCBULK (143)</b>      | LOCBULK=1 or 2 specifies that Bulk Data is being obtained via the DBLOCATE FMS statement. NASTRAN LOCBULK=2 is specified when no Bulk Data entries, except for PARAM entries, are to be deleted or added. All PARAM entries must be respecified. All other entries will be ignored and, if present, may increase CPU times in XSORT and IFP. With LOCBULK=2, the XSORT and IFP modules will not reprocess the Bulk Data Section stored in the SEMAP run. Also, GP1, TASNP2, SEP1 and SEP1X modules will be skipped. 0 is the default, which assumes the RESTART FMS statement. This system cell is recognized only in SOLs 101 through 200. |   |
| <b>(144)</b>              | RESTART FMS statement existence flag. Set to 1 if RESTART statement is present.   |   |
| <b>BFGS (145)</b>         | Selects strategies of BFGS updates for the arc-length methods in nonlinear analysis. Please see the <i>MSC Nastran Nonlinear Handbook</i> .   |   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |   |
|---------------------------|---|---|
|                           | 0   | Update $\Delta u_R$ and $\Delta u_p$ at every iteration with $\gamma^*$ . (Default) |
|                           | 1   | Update $\Delta u_R$ only with $\gamma$  |
|                           | 2   | Update $\Delta u_R$ only with $\gamma^*$ .  |
| FBSMEM (146)              | <p>Reserves part of Nastran open core memory for faster solution in the Lanczos method of eigenvalue extraction. Default = 0. When FBSMEM is less than or equal to 0, available memory is used to store the factor matrix for use during forward-backward substitution (FBS) operations (aka “factor caching”). System Information Message 4199 is printed in the F04 file when the factor matrix is cached.</p> <p>Set FBSMEM=1 to disable Lanczos factor caching.</p> |   |
| UWM (147)                 | <p>SYSTEM(147)=1 issues a User Warning Message for a DMAP parameter appearing on a CALL statement that has an inconsistent authorization in the called subDMAP. 0 is the default, which means no message is issued. A value of 1 also issues User Warning Message 1 during DMAP compilation if a module instruction is missing trailing commas for input and output data blocks.</p>  |   |
| DBVERCHK (148)            | <p>In general, databases are not compatible between major releases; therefore, a check is performed in MSC Nastran to ensure that the major version which created the database is the same as that being executed. Since specific data on the database may be compatible, SYSTEM(148) allows this check to be circumvented. However, circumventing the check may lead to problems later in the run.</p>   |   |
|                           | 0   | Check is performed (Default)  |
|                           | 1   | Check is not performed  |
| SCR300DEL (150)           | <p>Sets minimum number of blocks of SCR300 partition of SCRATCH DBset at which it is deleted. <a href="#">INIT</a>, <a href="#">98</a> FMS statement (Default = 100).</p>   |   |
| (151)                     | <p>Requests spill or no spill of the SCR300 partition of SCRATCH DBset. <a href="#">INIT</a>, <a href="#">98</a> FMS statement (Default = 0).</p>   |   |
| DBLAMKD (155)             | <p>Differential stiffness formulation for CBEAM and CTETRA elements.</p>  |   |
|                           | 0   | Current formulation (Default)   |
|                           | 1   | Pre-Version 67 formulation  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
|---------------------------|--|---|---|-----------|---|--|---|--------------------|---|--|---|--|----|--|----|--|------|--|
| (162)                     |  | <p>Reserves MSC Nastran executive first number for GINO for each of the tables DBTEMP, FCB, DBDICT and FIST. Default value is 6000 and is often controlled in DMAP just before and just after modules that deal with large input families. Usually a user will have no need to modify. However if the user gets the following message:</p> <p>*** SYSTEM FATAL MESSAGE 1267 (XSTORE)<br/>     NO MORE MEMORY SPACE AVAILABLE FOR EXECUTIVE TABLES.<br/>     USER ACTION: INCREASE DEFAULT NUMBER OF EXECUTIVE TABLES ENTRIES VIA SYSTEM CELL 162.<br/> <b>SYSTEM(162) IS CURRENTLY SET TO 6000</b></p> <p>A DMAP workaround is of the form:<br/>     Just before the offending module<br/>     NP=GETSYS(NP,162) \$<br/>     NP1= value \$ value &gt; 6000<br/>     PUTSYS(NP1,162) \$<br/>     Just after offending module<br/>     PUTSYS(NP,162) \$</p>  |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| (166)                     |  | <p>Controls sparse symmetric decomposition. Sum the desired values (Default = 0).</p> <table border="1"> <tr> <td>0</td><td>No action</td></tr> <tr> <td>1</td><td>If insufficient core is encountered, then switch to conventional decomposition and continue (Default).</td></tr> <tr> <td>2</td><td>Print diagnostics.</td></tr> <tr> <td>4</td><td>Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.</td></tr> <tr> <td>8</td><td>Output a matrix containing the maximum ratio vector in the output slot for the upper factor.</td></tr> <tr> <td>32</td><td>Turn off internal matrix scaling in the READ module.</td></tr> <tr> <td>64</td><td>Turn off internal matrix balancing in the READ module.</td></tr> <tr> <td>8192</td><td>Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information.</td></tr> </table> | 0 | No action | 1 | If insufficient core is encountered, then switch to conventional decomposition and continue (Default). | 2 | Print diagnostics. | 4 | Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules. | 8 | Output a matrix containing the maximum ratio vector in the output slot for the upper factor. | 32 | Turn off internal matrix scaling in the READ module. | 64 | Turn off internal matrix balancing in the READ module. | 8192 | Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information. |
| 0                         | No action  |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 1                         | If insufficient core is encountered, then switch to conventional decomposition and continue (Default).   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 2                         | Print diagnostics.   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 4                         | Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 8                         | Output a matrix containing the maximum ratio vector in the output slot for the upper factor.   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 32                        | Turn off internal matrix scaling in the READ module.   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 64                        | Turn off internal matrix balancing in the READ module.   |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |
| 8192                      | Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information. |   |   |           |   |  |   |                    |   |  |   |  |    |  |    |  |      |  |

Table 2-1      System Cell Summary (continued)

| System Cell Name (Number)      | Function and Reference  |  |
|--------------------------------|---|--|
| <a href="#">LDQRKD (170)</a>   | Selects the differential stiffness method for CQUAD4 and CTRIA3 elements:   |  |
|                                | 0   | Version 68, improved method (Default).                               |
|                                | 1   | Pre-Version 68 method  |
| <a href="#">OLDQ4K (173)</a>   | Requests the pre-Version 68 CQUAD4 element stiffness formulation. No value is required after the keyword. Equivalent to SYSTEM(173)=1.  |  |
|                                | 0   | Default  |
|                                | 1   | Requests pre-V68 QUAD4 formulation.                                  |
|                                | 2   | Requests V68 - V70.5 QUAD4 formulation.                              |
| <a href="#">Q4TAPER (189)</a>  | Specifies the maximum allowable value of taper for CQUAD4 element. Taper is computed by connecting opposite grid points and computing the area of the enclosed triangles. Another way to think of taper is the ratio of the areas on the two sides of a diagonal (Real $\geq 0.0$ ; Default = 0.5). |  |
| <a href="#">Q4SKEW (190)</a>   | Specifies the minimum allowable value of skew for the CQUAD4 element. Skew is the angle measured in degrees between the lines that join opposite midsides (Real $\geq 0.0$ ; Default = 30.0).   |  |
| <a href="#">TETRAAR (191)</a>  | Specifies the maximum allowable aspect ratio of the longest edge to the shortest altitude for the CTETRA element (Real $\geq 0.0$ ; Default=100.0).   |  |
| <a href="#">SCRSAVE (196)</a>  | Lanczos high performance option: Controls reuse of scratch files in segment logic.  |  |
|                                | 0   | Do not reuse (Default)   |
|                                | 1   | Reuse  |
| <a href="#">MINFRONT (198)</a> | Lanczos high performance option: minimum front size. (The default value is machine dependent).  |  |
| <a href="#">NSEGADD (200)</a>  | Number of segments in the element error table that is generated in adaptive analysis (Default = 2).   |  |
| <a href="#">CORDM (204)</a>    | Specifies the default value for CORDM field on the PSOLID entry (Integer $\geq -1$ ; Default = 0).  |  |
| <a href="#">(205)</a>          | Rank to use for real symmetric sparse decomposition high rank update. Default is hardware dependent.  |  |
| <a href="#">DCMPSEQ (206)</a>  | Selects ordering method for sparse matrix decomposition.  |  |
|                                | 0   | Method selected automatically in symbolic factoring phase (Default). |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
|                           | 1  | Minimum degree ordering.  |
|                           | 2  | Modified minimum degree ordering for indefinite matrices.   |
|                           | 3  | No ordering (uses given sequence).  |
|                           | 4  | Extreme ordering. (BEND)  |
|                           | 8  | METIS ordering. Metis was developed by George Karpis and Vipin Kumar at the University of Minnesota. More information may be found at <a href="http://www.cs.umn.edu/~karypis/metis">http://www.cs.umn.edu/~karypis/metis</a> . |
|                           | 9  | Selects the better of METIS and MMD.  |
|                           | 10   | Selects dof-based (rather than grid-based) modified minimum degree ordering.  |
|                           | 68   | This option may reduce the number of nonzero factors in the sparse decompensation method.   |
|                           | 132  | Similar to 68 but does not require the USET and SILS table as input. Uses extreme reordering.   |
|                           | 136  | Same as 132 but performs METIS reordering.  |
|                           | 260  | Selects dof-based (rather than grid-based) extreme ordering.  |
|                           | 264  | Selects dof-based (rather than grid-based) METIS ordering.  |
|                           | 1024   | Selects Minimum Degree reordering in Intel MKL Pardiso sparse solver.   |
|                           | 2048   | Selects METIS reordering in Intel MKL Pardiso sparse solver.  |
|                           | 4096   | Selects OpenMP (SMP) METIS reordering in Intel MKL Pardiso sparse solver.   |
| USPARSE (209)             | Unsymmetric sparse matrix method selection for the decomposition and forward-backward substitution operations. |   |
|                           | 0  | Deactivate  |
|                           | 1  | Select Nastran unsymmetric sparse factorization (Default).  |
|                           | 16   | Select UMFPACK unsymmetric sparse factorization.  |
|                           | 64   | Select LUSOL unsymmetric sparse factorization.  |
| PUNCHTYPE (210)           | Used to control punch formula.   |   |
|                           | 0  | “Old” punch, default in MSC Nastran 2001 and earlier versions.  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference |  |
|---------------------------|------------------------|--|
|                           | 1                      | “New” punch, default in MSC Nastran 2004 and uses the NDDL.  |
|                           | 2                      | Same as 1 except the line numbers are eliminated.  |
| CHEXAINT (212)            |                        | Specifies CHEXA element’s integration rule for p-adaptive analysis and p=2x2x2 (only).   |
|                           | 0                      | Reduced (Default)  |
|                           | 1                      | Full   |
| DISTORT (213)             |                        | Element distortion fatal termination override. Applies to all p-elements and the TETRA h-elements.   |
|                           | 0                      | Terminate run (Default)  |
|                           | 1                      | Do not terminate run.  |
| T3SKEW (218)              |                        | Allows the user to control the minimum vertex angle for CTRIA3 elements at which USER WARNING MESSAGE 5491 is issued. See the description of CTRIA3.                       |
| (219)                     |                        | Rank to use for complex symmetric sparse decomposition high rank update (Default = 1).   |
| (220)                     |                        | Rank to use for real unsymmetric sparse decomposition high rank update (Default = 1).  |
| (221)                     |                        | Rank to use for complex unsymmetric sparse decomposition high rank update (Default = 1).   |
| (253 - 262)               |                        | SYSTEM(252) to (262) have been set aside for user DMAPS. MSC will not use these values in its code in present or future versions. The SSSAlter library may use this range. |
| MAXSET (263)              |                        | Controls the default number of vectors in block or set for Lanczos Eigenvalue extraction. See EIGRL, 1924. The default is 7 for most machines but it is machine dependent. |
| QUARTICDLM (270)          |                        | A value of 1 selects the new quartic formulation of the doublet lattice kernel (N5KQ), while 0 selects the original quadratic form (Default = 0).                          |
| (273)                     |                        | A value of 1 selects the old Lanczos shift logic from Version 70 and previous systems (Default = 0).   |
| DBCFACT (274)             |                        | Option to create an xdb file with a multi-key data format and is intended for “large” xdb files.   |
|                           | 0                      | no multi-key format (default)  |
|                           | 2                      | auto-select multi-key format   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |  |
|---------------------------|---|--|
|                           | 4   | multi-key format   |
| (275)                     | Specifies the timeout for ISHELL in seconds. Values greater than 2,678,400 (31 days) will be set to 31 days.  |  |
| SPLINE_METRICS (281)      | Print additional spline quality information   |  |
|                           | =0  | Don't print (default).   |
|                           | =1  | Print quality information for each user-generated spline.  |
|                           | =2  | Print quality information for each user-generated spline and each Nastran-generated center-to-corner point spline. |
|                           | Note that if SPLINE_METRICS is defined on the NASTRAN statement without a numeric value, then a value of 1 will be used.  |  |
| MINDEF (303)              | Indefinite mass matrix check, the Default = 1 does not perform the check.   |  |
|                           | > 0   | Check is not performed   |
|                           | < 0   | Epsilon is set to -1.E(MINDEF)   |
|                           | 0   | MINDEF defaults to -6  |
| MPERTURB (304)            | Perturbation factor for indefinite mass matrix. The Default = 1 does not perturb the mass.  |  |
|                           | > 0   | The mass is not perturbed  |
|                           | < 0   | The mass 1.E(MPERTURB) is added to the diagonal terms of the mass matrix   |
|                           | 0   | MPERTURB defaults to -6. The perturbed mass matrix is used in the subsequent eigenvalue analysis                   |
| (309)                     | If set to 1, requests the pre-Version 70.7 CHEXA8 element stiffness formulation (Default = 0).  |  |
| OLDRBE3 (310)             | If set to 1, requests the pre-Version 70.7 RBE3 formulation (Default = 0).  |  |
| TBCMAG (311)              | Change the stiffness to 1.0E2 if using thermal conductivity in Btu/sec/in.F. See Bulk Data entry TEMPBC, 3226 for more information. The default stiffness is 1.0E10.  |  |
| INDEX (316)               | Indexes and/or saves a minimum set of data blocks to the database needed to for postprocessing in Patran or the toolkit. This cell must be used with scratch=no on the nastran command. This cell has several options which are set by adding the following values: |  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |  |
|---------------------------|---|--|
|                           | 1   | Index IFP data blocks.   |
|                           | 2   | Index OFP data blocks.   |
|                           | 4   | Save above data blocks to the MASTER dbset.  |
|                           | 16  | Save above data blocks to the DBALL dbset.   |
|                           | For example, INDEX=7 will index the IFP and OFP data blocks and save them to the MASTER dbset. (Scratch=post is equivalent to scratch=no and INDEX=19.) But with INDEX>0 the resulting database is not restartable. |  |
| XMSG (319)                | If set to 1, gives extended error messages (Default = 0).   |  |
| OLDDAREA (320)            | Do not convert DAREA Bulk Data entries for grid and scalar points to equivalent FORCE/MOMENT/SLOAD Bulk Data entries (equivalent to SYSTEM(320) = -1).  |  |
|                           | Controls the conversion of DAREA Bulk Data entries for grid and scalar points to equivalent FORCE/MOMENT/ SLOAD Bulk Data entries as appropriate.   |  |
|                           | 0   | Perform the conversion, but do not give details of the conversion (Default).   |
|                           | N   | Perform the conversion and give details of the first N such conversions.   |
|                           | -1  | Do not perform the conversion.   |
| RSEQCONT (357)            | If system cell IFPSTAR=NO, RSEQCONT default = 0   |  |
|                           | 1   | causes all continuation fields to be ignored and treated as blank. If set to 1, the continuation entries must immediately follow the parent  |
|                           | 2   | causes fatal message to be issued if there are orphaned continuation entries.  |
|                           | System cell (357) only works when IFPSTAR=NO, it is fixed to 1 when IFPSTAR=YES   |  |
| QLHOUL (359)              | 0   | Use the user-requested eigensolution method.   |
|                           | < > 0   | When LAN is requested, switch to AHOU if the number of DOFs sent to the eigensolver is ≤ “nswitch”, an input parameter to the READ module. This parameter has an MPL default of 20. It may be set to other values in the solution sequences, depending on the context. When HOU, MHOU, or AHOU is selected, switch to the new Householder-QL solution (Default = 1). |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number)       | Function and Reference   |  |     |   |   |   |     |   |
|---------------------------------|--|--|-----|---|---|---|-----|---|
| <a href="#">PRTPCOMP (361)</a>  | If set to 1, then the equivalent PSHELLs and MAT2s from PCOMPs or PCOMPGs are printed to the .f06 file provided that ECHO = NONE is not set. (Default = 0, suppresses this printout).  |  |     |   |   |   |     |   |
| <a href="#">STRICTUAI (363)</a> | A value of 1 accepts strict UAI/Nastran Bulk Data entries (Default = 0).   |  |     |   |   |   |     |   |
| <a href="#">STPFLG (366)</a>    | Selects the SUBCASE or STEP layout when there are a number of SUBCASE commands and no STEP command in a Case Control file for SOL 400 (Default = 0). <table border="1" style="margin-left: 20px;"> <tr> <td>0</td> <td>Keep all SUBCASE commands in the Case Control file and insert a "STEP 1" for each SUBCASE.</td> </tr> <tr> <td>1</td> <td>Convert all the SUBCASE IDs into STEP IDs, and then insert a "SUBCASE 1" before the first STEP.</td> </tr> </table>   |  | 0   | Keep all SUBCASE commands in the Case Control file and insert a "STEP 1" for each SUBCASE.                              | 1 | Convert all the SUBCASE IDs into STEP IDs, and then insert a "SUBCASE 1" before the first STEP. |     |   |
| 0                               | Keep all SUBCASE commands in the Case Control file and insert a "STEP 1" for each SUBCASE.   |  |     |   |   |   |     |   |
| 1                               | Convert all the SUBCASE IDs into STEP IDs, and then insert a "SUBCASE 1" before the first STEP.  |  |     |   |   |   |     |   |
| <a href="#">QRMETH (370)</a>    | When the input consists of CQUADR/CTRIAR elements, QRMETH allows you to convert them to QUAD4/TRIA3 elements or when the input consists of QUAD4/TRIA3 elements to convert them to CQUADR/CTRIAR elements.   |  |     |   |   |   |     |   |
|                                 | Some of the output requests for CQUAD4/CTRIA3 are not available for CQUADR/CTRIAR, for example, the CUBIC option on the STRESS Case Control command. In this case, the equivalent CQUADR/CTRIAR options are used. <table border="1" style="margin-left: 20px;"> <tr> <td>0</td> <td>Use the user input CQUADR/CTRIAR elements. (Default)</td> </tr> <tr> <td>3</td> <td>Converts user input CQUADR/CTRIAR into CQUAD4/CTRIA3.</td> </tr> <tr> <td>5</td> <td>Converts user input CQUAD4/CTRIA3 into CQUADR/CTRIAR.</td> </tr> </table> |  | 0   | Use the user input CQUADR/CTRIAR elements. (Default)  | 3 | Converts user input CQUADR/CTRIAR into CQUAD4/CTRIA3.   | 5   | Converts user input CQUAD4/CTRIA3 into CQUADR/CTRIAR. |
| 0                               | Use the user input CQUADR/CTRIAR elements. (Default)   |  |     |   |   |   |     |   |
| 3                               | Converts user input CQUADR/CTRIAR into CQUAD4/CTRIA3.  |  |     |   |   |   |     |   |
| 5                               | Converts user input CQUAD4/CTRIA3 into CQUADR/CTRIAR.  |  |     |   |   |   |     |   |
| <a href="#">PARAMCHK (372)</a>  | DMAP parameter initialization check. <table border="1" style="margin-left: 20px;"> <tr> <td>0</td> <td>Issue User Fatal Message for an input parameter that is not used in a type statement in the subDMAP argument (Default).</td> </tr> <tr> <td>1</td> <td>Issue User Fatal Message for the initialized parameter.</td> </tr> </table>  |  | 0   | Issue User Fatal Message for an input parameter that is not used in a type statement in the subDMAP argument (Default). | 1 | Issue User Fatal Message for the initialized parameter.   |     |   |
| 0                               | Issue User Fatal Message for an input parameter that is not used in a type statement in the subDMAP argument (Default).  |  |     |   |   |   |     |   |
| 1                               | Issue User Fatal Message for the initialized parameter.  |  |     |   |   |   |     |   |
| <a href="#">TZEROMAX (373)</a>  | Controls time step adjustment in nonlinear transient analysis. <table border="1" style="margin-left: 20px;"> <tr> <td>&gt; 0</td> <td>Maximum number of times to return to time zero.</td> </tr> <tr> <td>0</td> <td>No initial time step adjustment (identical to V2001), default for SOL 400 with CGAP elements.</td> </tr> <tr> <td>&lt; 0</td> <td>No limit on DT adjustment.</td> </tr> </table>  |  | > 0 | Maximum number of times to return to time zero.   | 0 | No initial time step adjustment (identical to V2001), default for SOL 400 with CGAP elements.   | < 0 | No limit on DT adjustment.                            |
| > 0                             | Maximum number of times to return to time zero.  |  |     |   |   |   |     |   |
| 0                               | No initial time step adjustment (identical to V2001), default for SOL 400 with CGAP elements.  |  |     |   |   |   |     |   |
| < 0                             | No limit on DT adjustment.   |  |     |   |   |   |     |   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |  |  |
|---------------------------|---|--|--|
|                           | 4   | Default for SOL 400 without CGAP element or for SOL 129. |  |
| <b>NOLIN (386)</b>        | (Real, Default = 1.0) Tolerance value for controlling adaptive time step bisection for NOLIN1 entries in SOL 129. Turned off for rotor dynamics. For NLRGAP see SYSTEM (431). |  |  |
|                           | 0.  | Bisection is suppressed (same as Version 2001).          |  |
|                           | .001  | Increase accuracy slightly.                              |  |
|                           | .01   | Increase accuracy a little more.                         |  |
|                           | 1.0   | Allow full adaptive time step bisection.                 |  |
| <b>KRYLOV1 (387)</b>      | Fast direct frequency response option.  |  |  |
|                           | -1  | Yes  |  |
|                           | 0   | No (Default)   |  |
| <b>KRYLOV2 (388)</b>      | Options related to fast direct frequency response analysis. Selects subspace generation method.   |  |  |
|                           | 1   | Lanczos (Default)  |  |
|                           | 2   | Arnoldi (Use for unsymmetric systems)                    |  |
| <b>KRYLOV3 (389)</b>      | Options related to fast direct frequency response analysis. Defines exponent of relative accuracy.  |  |  |
|                           | -4  | Error<1.0E-4 (Default)                                   |  |
|                           | -6  | Error<1.0E-6   |  |
| <b>KRYLOV4 (390)</b>      | Options related to fast direct frequency response analysis. Defines pole selection distance.  |  |  |
|                           | 0   | Next pole is next un converged frequency (Default).      |  |
|                           | 2   | 2*next frequency distance.                               |  |
| <b>KRYLOV5 (391)</b>      | Options related to fast direct frequency response analysis. Selects decomp/fbs tradeoff parameter determining Krylov method accuracy.   |  |  |
|                           | 0   | never terminate, do FBS iterations till accuracy is met  |  |
|                           | 1   | terminate when FBS time > decomposition time (Default)   |  |
|                           | 2   | terminate when FBS time > 2 * decomposition time         |  |
|                           | -2  | terminate when FBS time > (1/2) * decomposition time     |  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |  |
|---------------------------|--|--|
| BARMASS (398)             | Allows the user to select the bar torsional mass moment of inertia. If set to 0, request the pre-MSC Nastran 2004 (Default = 0). If set to greater than 0, the torsional mass moment of inertia term is included in the mass matrix formulation of bar elements. For both values of COUPMASS, the torsional inertia is added. For COUPMASS = 1, the axial mass will be consistent rather than coupled. |  |
| DPBLKTOL (402)            | Specifies Bulk Data tolerance value for GRID, CORD2C, CORD2R, and CORD2S entries. (Default = 0). See / Control Input Stream section under Bulk Data Entries below  |  |
|                           | < 0  | Do not remove duplicate entries  |
|                           | 0.0  | Check specified Bulk Data entries for exact physical match and remove duplicates   |
|                           | > 0  | Then perform the DPBLKTOL=0.0 check and additionally GRID entry as duplicate if $\{ x1(i) - x2(i)  \leq DPBLKTOL ; i = 1, 2, 3 \text{ and } (cp1 \neq cp2 \text{ and } cp1 \cdot cp2 = 0) \text{ and } (cd1 \neq cd2 \text{ and } cd1 \cdot cd2 = 0) \text{ and } (ps1 = ps2) \text{ and } (seid1 = seid2)\}$ using entry with $cpi \neq 0$ and $cdi \neq 0$ if possible |
| OP2NEW (403)              | Selects the additional version information in the OUTPUT2 file. (Default = 0)  |  |
|                           | 0  | Leave alone and unidentified, pre-2004 convention  |
|                           | 1  | Add version major, minor, special to tape label and change IFP datablock locate code word 3  |
| DEF_DENS (408)            | Set DEFAULT value for the MODEL_CHECK Executive statement MAT_DENSITY=DEFAULT operation (Default = 0.0).   |  |
| DEF_TECO (410)            | Set DEFAULT value for the MODEL_CHECK Executive statement MAT_TECO=DEFAULT operation (Default = 0.0).  |  |
| DEF_TEIJ (411)            | Set DEFAULT value for the MODEL_CHECK Executive statement MAT_TEIJ=DEFAULT operation (Default = 0.0).  |  |
| DEF_DAMP (412)            | Set DEFAULT value for the MODEL_CHECK Executive statement MAT_DAMP=DEFAULT operation (Default = 0.0).  |  |
| OPTCOD (413)              | Specifies which optimization code to be used in SOL 200 (Default = 0; automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory).   |  |
|                           | Note: Options 1 and 2 are no longer used.  |  |
|                           | 3  | MSCADS   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference |  |
|---------------------------|------------------------|--|
|                           | 4                      | IPOPT Optimizer  |
| (414)                     |                        | Only used if COUPMASS < 0. Default = 0 Coupled Mass for CBAR and CBEAM elements.   |
|                           | > 0                    | Lumped Mass matrix will contain translational components only for CBAR and CBEAM elements.   |
| OLDTLDMTH (428)           |                        | If nonzero, requests the pre-version 2005 r3 method for computing thermal expansion in CHEXA, CPENTA, and CTETRA elements.   |
| NONLRGAP (431)            |                        | (Real, Default = -1.0) Sets tolerance value for controlling adaptive time step bisection for NLRGAP entries in SOL 129 and SOL 400. Time step bisection will occur if the contact force magnitude changes more than the tolerance. |
|                           | = 0.                   | (Same logic as Version 2005)   |
|                           | < 0.                   | Tolerance = 1.E+9  |
|                           | > 0.                   | Tolerance = 1000./SYSTEM(431)  |
| ESLNRO (443)              |                        | Flag to invoke Nonlinear Response Optimization with the concept of Equivalent Static Loads   |
|                           | 0                      | No ESLNRO (Default)  |
|                           | 1                      | Turn on ESLNRO   |
| (444)                     | 0                      | Selects the pre Nastran 2010 release of IFP for bulk data processing.  |
|                           | 1                      | Selects the MSC Nastran 2010 release Common Data Model for data processing, do not allow an integer in a field that the only possible format is real. (Default)  |
|                           | 9                      | Selects the MSC Nastran 2010 release Common Data Model for data processing, allow an integer in a field that the only possible format is real.   |
| IFPSTAR                   | YES                    | Selects the MSC Nastran 2010 release Common Data Model for data processing, do not allow an integer in a field that the only possible format is real. (Default)  |
|                           | NO                     | Select the MSC Nastran IFP for bulk data processing.   |
|                           |                        | On command line IFPSTAR=YES or NO is able to express the value of 1 or 0 only.   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number)   | Function and Reference   |   |
|---|--|---|
| MNLQ4C (445)  | Allows or disallows corner stress calculations for CQUAD4 if material nonlinear (Default = 1). |   |
|   | 0  | Allows material nonlinear CQUAD4 corner stress calculations if plastic deformation has occurred, results may be totally incorrect.  |
|   | 1  | Disallows material nonlinear CQUAD4 corner stress calculations. If corner requested and material nonlinear sensed, corner is turned off and center only is computed.  |
| (446)   | Controls 4 noded CTETRA fluid mass calculations. (Default=0)                                   |   |
|   | 0  | To improve the modal results for fluid mass and remove the possibility of spurious fluid modes, a four point integration method is used.  |
|   | 1  | A one point integration method is used to compute the fluid mass.   |
| PARSLAVE(449)   | Bitwise control of DMP operations in SOL 400 nonlinear analysis.                               |   |
| Bit   | Value  | Description   |
| 1   | Default  | System determines automatically at job startup. When running the job on multiple hosts, the value is set to 0; otherwise (single host), it is set to 1. <b>User setting of PARSLAVE is not recommended.</b> |
|   | 0  | Slave processes execute the entire analysis. This value is recommended for distributed execution.   |
|   | 1  | Slave processes only participate in parallel computational tasks. Recommended for DMP running on a single host (i.e., non-distributed DMP execution).   |
| 2   | 0  | DMP linear equation solver disabled.  |
|   | 1  | DMP linear equation solver enabled (default).   |
| 3   | 0  | DMP nonlinear element matrix generation disabled.   |
|   | 1  | DMP nonlinear element matrix generation enabled (default).  |
| To change defaults for this system cell, use the DOMAINsolver command in the <i>MSC Nastran Quick Reference Guide</i> . |  |   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |            |  |            |   |           |  |           |   |
|---------------------------|--|------------|--|------------|---|-----------|--|-----------|---|
| (451)                     | If set nonzero, requests the method to calculate transverse shear correction for elements using MID4, or "Z0" on the PCOMP to offset a shell element. The default 0 uses a new method that avoids excessive transverse shear stiffness. This is expected to have small effect on most models, unless the shell is thick and the offset is large.   |            |  |            |   |           |  |           |   |
| DIFFS(614)                | Flag to control differential stiffness computation (Default=0). <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding: 5px;">DIFFS = -2</td><td style="padding: 5px;">Enforce differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.</td></tr> <tr> <td style="padding: 5px;">DIFFS = -1</td><td style="padding: 5px;">Skip differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.</td></tr> <tr> <td style="padding: 5px;">DIFFS = 0</td><td style="padding: 5px;">Compute differential stiffness matrix. For nonlinear elements with geometric nonlinear analysis in SOL 400, also compute follower force stiffness. For linear buckling analysis, differential stiffness calculations for RBE3 elements will be skipped when the maximum number of grids exceeds 10000.</td></tr> <tr> <td style="padding: 5px;">DIFFS &gt; 0</td><td style="padding: 5px;">Defined via Bulk Data entry MDLPRM,NLDIFF to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis in SOL 400.</td></tr> </table> | DIFFS = -2 | Enforce differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids. | DIFFS = -1 | Skip differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids. | DIFFS = 0 | Compute differential stiffness matrix. For nonlinear elements with geometric nonlinear analysis in SOL 400, also compute follower force stiffness. For linear buckling analysis, differential stiffness calculations for RBE3 elements will be skipped when the maximum number of grids exceeds 10000. | DIFFS > 0 | Defined via Bulk Data entry MDLPRM,NLDIFF to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis in SOL 400. |
| DIFFS = -2                | Enforce differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.   |            |  |            |   |           |  |           |   |
| DIFFS = -1                | Skip differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.  |            |  |            |   |           |  |           |   |
| DIFFS = 0                 | Compute differential stiffness matrix. For nonlinear elements with geometric nonlinear analysis in SOL 400, also compute follower force stiffness. For linear buckling analysis, differential stiffness calculations for RBE3 elements will be skipped when the maximum number of grids exceeds 10000.   |            |  |            |   |           |  |           |   |
| DIFFS > 0                 | Defined via Bulk Data entry MDLPRM,NLDIFF to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis in SOL 400.  |            |  |            |   |           |  |           |   |
| IFPBUFF(624)              | Specifies the size of IFPSTAR data base I/O transfers<br><br>Default value=1024 words<br>The physical I/O size is IFPBUFF words. The maximum value of IFPBUFF is 65536 words.<br><br>Example: NASTRAN IFPBUFF=8192   |            |  |            |   |           |  |           |   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
| NONUPIV (653)             | Parameter to select the numeric compute kernel and pivoting methods in MSCLDL and MSCLU sparse direct solvers. Also see the bulk data entry <a href="#">MDLPRM, 2452</a> ,NONUPIV.   |   |
|                           | 0  | Use the native Bunch-Kauffman threshold pivoting in MSCLDL, and the native threshold partial pivoting in MSCLU (Default).   |
|                           | 1  | Use no numeric pivoting in MSCLDL and MSCLU. BLAS3 TRSMs are called to compute the pivot column update to improve performance. Ill-conditioned models may die of “singular matrix” during sparse factorization. |
|                           | 3  | LAPACK SYTRFs with Bunch-Kaufman pivoting and GETRFs with partial pivoting are called to perform factorizations, and BLAS3 TRSMs are called to compute pivot column update to improve performance.              |
| GPU_MIN_RANK (655)        | If the rank of the frontal matrix is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. The default value for GPU_MIN_RANK is 32.   |   |
| GPU_MIN_FRONT (656)       | The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via gpu_min_front and its companion parameter, gpu_min_rank. The value specified must be an integer greater or equal to 1. |   |
|                           | If the front size is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. The default value for GPU_MIN_FRONT is 2048.  |   |
| (662)                     | Controls CBUSH internal load calculation for radial dependence. (Default=0)  |   |
|                           | 0  | Radial dependence of PBUSHT entries DOES NOT depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.   |
|                           | 1  | Radial dependence of PBUSHT entries DOES depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
| SEGLOG(664)               | Control the logic to compute the default value of PENALT in segment-to-segment contact analysis. Before v2017, Default value is 0. |   |
|                           | 0  | Use the existing/old logic where the default value is based on the average elastic stiffness of the two contact bodies and the default error tolerance.   |
|                           | 1  | Use new logic (Default) for PENALT to avoid a too large value of PENALT, where the default value is defined by the body with the softer material and the minimum edge length of the two contact bodies. This logic is often better for thin shell structures. |
| CNTBKCMP(666)             | Control the logic to compute the contact at shell corners in node-to-segment contact analysis to ensure backward compatibility.    |   |
|                           | 0  | Use the current/new contact logic for shell corners (Default). This logic uses weighted average normal vector and adjusted thickness for a better representation of the sharp corner at both top and bottom of the shell.                                     |
|                           | 1  | Use old contact logic for shell corners to ensure backward compatibility. This logic uses average thickness and average normal vector even at sharp corners.  |
| (670)                     | Control for Automatic Contact Generation. Used when BCONTACT=AUTO is present in case control.                                      |   |
|                           | 0  | Default, Nastran run to completion after generating acg file.   |
|                           | 3  | Nastran terminates after generating acg file.   |
| LMFBKCMP(676)             | Control the logic to compute the default value of Lagrange multiplier (LMFACT and PENFN)   |   |
|                           | 0  | (Default) Use new algorithm for LMFACT/PENFN, which are two order lower than previous value used before 2013.0 Version.   |
|                           | 1  | Use old algorithm used before 2013.0 Version.   |
| FF0BKCMP(677)             | Control the logic to compute the unloading follower force.   |   |
|                           | 0  | (Default) Use new algorithm for unloading follower force, i.e., follower force in the previous loadcase will be unloaded by follower force method but not by linear interpolation.  |
|                           | 1  | Use old algorithm for unloading follower force, i.e., by linear interpolation (before 2013.0 Version).  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
| <b>OLDLCNTR(678)</b>      | Ensure backward compatibility for linear connector elements.   |   |
|                           | 0  | Use existing (after V2014.1) algorithm with RBE3 relationships for linear connectors (Default).   |
|                           | 1  | Use old algorithm (V2014.1 or earlier) for linear connectors to ensure backward compatibility.  |
|                           | PARAM,OLDWELD,YES is equivalent to a value NASTRAN OLDLCNTR=1  |   |
| <b>(679)</b>              | Controls checking for validity of sparse matrix passed to Intel MKL Pardiso. Column indices must be in increasing order within rows. |   |
|                           | 0  | Disable matrix checking (Default)   |
|                           | 1  | Enable matrix checking  |
| <b>(684)</b>              | Control of matrix type for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Parido documentation</a>               |   |
| <b>RDBOTH(695)</b>        | Parameter to select Rayleigh damping approach for rotordynamics (compatibility with V2005) implementation, Integer                   |   |
|                           | 0  | Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)  |
|                           | 1  | Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT are not used |
|                           | 2  | Ignore circulation effects in rotordynamic analysis.  |
|                           | 4  | Include effect of stress stiffening using method = 1 (see RFORCE entry)   |
|                           | 8  | Include effect of stress stiffening using method = 2 (see RFORCE entry)   |
| <b>S2SDEF(701)</b>        | Parameter to control backward compatibility of Seg-to-Seg contact default values.  |   |
|                           | 0  | Original Seg-to-Seg contact default values  |
|                           | 1  | Using new Seg-to-Seg contact default values in and after 2016 version. (Default)  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |  |
|---------------------------|---|--|
| HDF5 (702)                | Control NH5RDB database creation.   |  |
| -1                        | Do not create NH5RDB database (Default).  |  |
| 0                         | Create NH5RDB database without compression.   |  |
| 1                         | Create NH5RDB database with compression.  |  |
| 2                         | Create uncompressed NH5RDB database without input data.   |  |
| 3                         | Create compressed NH5RDB database without input data.   |  |
| OBEAMS (725)              | Control Linear format output of Advanced Nonlinear Bar Element.<br>Default: 0   |  |
| 0                         | New. Output of beam stress is separated into Bending and Axial stresses.  |  |
| 1                         | Old. Output of beam stress combines bending and axial stresses together.  |  |
| OLDCCONE (726)            | The CCONEAX Force/Stress Recovery element coordinate system is inconsistent with the input RINGAX coordinate system relative to the RINGAX A/RINGAX B order. The CCONEAX Force/Stress Recovery has now been made consistent. For those who may have post processing that recognized the inconsistency and do not wish to change, the inconsistency may be restored with OLDCCONE. |  |
| 0                         | Default: Force/Stress output consistent with RINGAX coordinate.   |  |
| 1                         | Restore inconsistent RINGAX A/RINGAX B Force/Stress output.   |  |
| METS2S (727)              | Control Segment to Segment contact.<br>Default: 0   |  |
| 0                         | Support SEGTOSEG only since 2017 BETA and later version   |  |
| 1                         | Old. Support SEGSMALL, SEGLARGE, as well SEGTOSEG.  |  |
| DMP (728)                 | The number of Distributed Memory Processes used. This option is only useful for specific Solution Sequences as described in the <i>MSC Nastran HPC User's Guide</i>   |  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number)     | Function and Reference  |   |
|-------------------------------|---|---|
| <a href="#">N2SDOF (729)</a>  | Control node freedom number of contact<br><br>Default: 0  |   |
|                               | 0   | Depend on contact type: 3 for touching contact; 6 for glued contact                               |
|                               | 1   | Set the node freedom of contact to be 6 always  |
| <a href="#">H5NORDOF(730)</a> | To suppress output rotational components to NH5RDB database   |   |
|                               | 0   | Output both translational and rotational components to NH5RDB database (Default).                 |
|                               | 1   | Do not output rotational components to NH5DB database.<br><br>Only used when HDF5(702) is 0 or 1. |
| <a href="#">H5MTX (739)</a>   | Write matrix data in separate file  |   |
|                               | 1   | Write matrix data into a separate file  |
|                               | 0   | Do not write matrix data into a separate file (Default)   |
| <a href="#">H5MDL (740)</a>   | Write model input data in separate file   |   |
|                               | 1   | Write model input data into a separate file   |
|                               | 0   | Do not write model input data into a separate file (Default)                                      |
| <a href="#">ACCSDLSZ(747)</a> | SDL work area memory size of IFPStar, default=100000<br><br>> 100000, work area size in word  |   |
|                               | Write GEOM3 and GEOM4 data in NH5RDB  |   |
|                               | -1  | Use OP2GM34 setting (Default).  |
| <a href="#">H5GM34 (751)</a>  | 1   | Write GEOM3 and GEOM4 data in NH5RDB  |
|                               | 0   | Do not write GEOM3 and GEOM4 data in NH5RDB   |
|                               | <a href="#">UDEFGRID(753)</a><br><br>Control whether having undefined grid points on ASET/ASET1 entries is allowed.<br><br>Default=0. |   |
|                               | 0   | Issue a User Warning Message and continue the simulation.   |
|                               | 1   | Issue a User Fatal Message and terminate the simulation   |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference   |   |
|---------------------------|--|---|
| CNTBACK(758)              | <p>BIAS=0.0 on BCONPRG/BCTABLE will be used as 0.0 in analysis.</p> <p>Before 2018.2, BIAS=0.0 is treated as default, same as blank, which will turn to the value of BIAS on BCPARA and so the user has to put a small value to approximate absolute 0.0.</p> <p>Since 20190.0, seg-to-seg initial stress free contact can adjust geometry to clear gap/penetration.</p> <p>Default=0.</p> |   |
|                           | 0  | BIAS=0.0 on BCONPRG/BCTABLE is treated as 0.0 and will override BIAS on BCPARA Seg-to-seg contact will adjust geometry by initial stress free, same as node-to-seg. |
|                           | 1  | (backward compatible) BIAS=0.0 on BCONPRG/BCTABLE is treated as blank; BIAS on BCPARA will be used in analysis.   |
|                           | 2  | Switch to Step Glue if Permanent Glue with LGDISP>1   |
|                           | 4  | (back to 2018.2) No geometry adjustment in seg-to-seg Contact Initial Stress Free.  |
| OP2NEW(761)               | <p>Select option to improve OUTPUT2 performance when converting data from I8 to I4 binary format.</p> <p>Default=1</p>   |   |
|                           | 0  | Use the old (till V2018.2) I8-I4 conversion method.   |
|                           | 1  | Use the new conversion method   |
| IGNBLN(767)               | <p>Control if ignore blank lines in an entry followed by continuation lines.</p>   |   |
|                           | 0  | (Default) Don't ignore and issue UFM when blank lines followed by continuations are found.  |
|                           | 1  | Ignore those blank lines.   |
| BLNMEM(768)               | <p>Memory size in bytes to store blank line numbers in input files.</p>  |   |
|                           | 8000:  | Default, 8000 bytes.  |
|                           | >=800:   | User specified size.  |
|                           | <800:  | Will be forcedly changed to 800.  |

Table 2-1 System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference  |  |           |   |           |  |
|---------------------------|---|--|-----------|---|-----------|--|
| OPENFSI(776)              | <p>Type: Integer</p> <p>OpenFSI interface option.</p> <table border="1"> <tr> <td>0</td><td>(Default) For implicit type, skip the putWettedNodeDisp method call when the current time is converged and no displacement update from last iteration.</td></tr> <tr> <td>1</td><td>Do not skip the putWettedNodeDisp call even there is no displacement update.</td></tr> </table> <p>Refer to <i>MSC Nastran User Defined Services User's Guide</i> for details.</p>  |  | 0         | (Default) For implicit type, skip the putWettedNodeDisp method call when the current time is converged and no displacement update from last iteration.        | 1         | Do not skip the putWettedNodeDisp call even there is no displacement update.   |
| 0                         | (Default) For implicit type, skip the putWettedNodeDisp method call when the current time is converged and no displacement update from last iteration.  |  |           |   |           |  |
| 1                         | Do not skip the putWettedNodeDisp call even there is no displacement update.  |  |           |   |           |  |
| H5INFO(789)               | <p>Write job run information in NH5RDB or not.</p> <table border="1"> <tr> <td>1</td><td>Yes (Default)</td></tr> <tr> <td>0</td><td>No</td></tr> </table>   |  | 1         | Yes (Default)   | 0         | No   |
| 1                         | Yes (Default)   |  |           |   |           |  |
| 0                         | No  |  |           |   |           |  |
| STRNCUR(778)              | <p>In nonlinear <u>strain recovery</u> for the CQUAD4 and CTRIA3 elements, especially for nonlinear composites with offsets and MDLPRM OFFDEF LROFF, an addition final pass of updating midplane strain and curvature is made. In some cases, this results in strain curvature results significantly different than those used to obtain the force and stress results. STRNCUR allows the user to obtain the midplane strain and curvature, and hence the ply strains, that are identical to the midplane strain and curvature used in force and stress recovery operations.</p> <table border="1"> <tr> <td>STRNCUR=0</td><td>Default: geometric nonlinear midplane strain &amp; curvature are updated for strain recovery and may not be identical to those used in force and stress recovery.</td></tr> <tr> <td>STRNCUR=1</td><td>Geometric nonlinear midplane strain &amp; curvature recovery values are the same values as used for element force and stress recovery.</td></tr> </table> |  | STRNCUR=0 | Default: geometric nonlinear midplane strain & curvature are updated for strain recovery and may not be identical to those used in force and stress recovery. | STRNCUR=1 | Geometric nonlinear midplane strain & curvature recovery values are the same values as used for element force and stress recovery. |
| STRNCUR=0                 | Default: geometric nonlinear midplane strain & curvature are updated for strain recovery and may not be identical to those used in force and stress recovery.   |  |           |   |           |  |
| STRNCUR=1                 | Geometric nonlinear midplane strain & curvature recovery values are the same values as used for element force and stress recovery.  |  |           |   |           |  |

Table 2-1      System Cell Summary (continued)

| System Cell Name (Number) | Function and Reference                                   |  |
|---------------------------|--|--|
| CNT101(786)               | Set Linear Contact (LINCNT=1) as default in SOL 101 only |  |
|                           | 0  | (Default) Linear Contact off/on depends on LINCNT on BCPARA<br>Note that Linear Contact is off (LINCNT=0) by default when no BCPARA entry present. |
|                           | 1  | Linear Contact is turned on in SOL 101, regardless of LINCNT on BCPARA.  |
| H5XHH (790)               | Write BHH, MHH and KHH matrices in NH5RDB                |  |
|                           | 0  | Do not write BHH, MHH and KHH in NH5RDB (Default)  |
|                           | 1  | Write BHH, MHH and KHH in NH5RDB   |

# 3

## File Management Statements

- Key to Descriptions
- The File Management Section (FMS)

## Key to Descriptions

### RESTART

Reuse Database From a Previous Run

A brief sentence about the function of the statement is given.

Requests that data stored in a previous run be used in the current run.

#### Format:

RESTART [ PROJECT = 'project-ID' VERSION = {version-ID} LAST ] { KEEP } { NOKEEP }

#### Describers:

project Brackets [] indicate that a choice of describers is optional. See PROJ FMS statement. Must be enclosed in single quotes, maximum of 8 characters, and must be preceded by a space. Braces {} indicate that a choice of describers is mandatory.

version-ID Version number. (Integer > 0).

LAST Specifies the last version under VERSION.

KEEP Data stored under VERSION will be retained after the run is completed.

NOKEEP Data stored under VERSION will be deleted from the database after the run is completed.

#### Remarks:

1. There may be only one RESTART statement in a run.
2. A new version-ID is automatically assigned whenever a restart is performed.
3. If project-ID or version-ID or both are specified and cannot be found a User Fatal Message will be issued.
4. The RESTART statement can be part of a sequence of statements (101 to 109). The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the describers.
5. If PROJECT is not specified on the PROJ statement. (See Example 2 below.)

#### Examples:

1. RESTART VERSION=7

Version number 7 will be retrieved for this run (version 8). At the end of the run version 7 will be deleted.

2. PROJ='FENDER'  
RESTART

The last version under project-ID FENDER will be used in the current run.

If the describers are stacked vertically, then only one may be specified.

The default describers are shaded.

Each of the describers is discussed briefly. Further details may be discussed under Remarks.

If the describer is in lower case, then it is a variable and the describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the describers.

## The File Management Section (FMS)

The File Management Section (FMS) is primarily intended for the attachment and initialization of Database sets (DBsets) and FORTRAN files. The initialization of DBsets includes specification of their maximum size, member names, and physical filenames. The initialization of FORTRAN files includes the specification of their filenames, FORTRAN unit numbers, and FORTRAN attributes.

In most classes of problems that use MSC Nastran solution sequences (SOLs), no File Management statements are required because a default File Management Section is executed at the beginning of every run. The default File Management Section is described in the *Database Concepts in the MSC Nastran Reference Guide*. If a restart is desired, then the RESTART statement is required. All other solutions may not be restarted. If the problem is large in terms of requiring significant amounts of memory or disk space, then the INIT, ASSIGN, and EXPAND statements may be required. If any FORTRAN files are required, then the ASSIGN statement is required; for example, the OUTPUT2 DMAP module. The ASSIGN statement is also required to assign databases for DBLOCATE, DBLOAD, and DBUNLOAD. Special database operations are performed by the DBLOCATE, DBLOAD, DBUNLOAD, DBLCLEAN, ACQUIRE, DBDICT, DBFIX, DBSETDEL, DBUPDATE, and PROJECT statements.

### File Management Statement Summary

The following is a summary of all File Management statements:

|          |  |
|----------|--|
| \$       | Comment statement.   |
| ACQUIRE  | Selects NDDL schema and MSC Nastran delivery database.             |
| ASSIGN   | Assigns physical files to DBset members or special FORTRAN files.  |
| CONNECT  | Group geometry data by evaluator and database.                     |
| DBCLEAN  | Deletes selected database version(s) and/or projects.              |
| DBDICT   | Prints the database directory in user-defined format.              |
| DBFIX    | Identifies and optionally corrects errors found in the database.   |
| DBLOAD   | Loads a database previously unloaded by DBUNLOAD.                  |
| DBLOCATE | Obtains data blocks and parameters from databases.                 |
| DBSETDEL | Deletes DBsets.  |
| DBUNLOAD | Unloads a database for compression, transfer, or archival storage. |
| DBUPDATE | Specifies the time between updates of the database directory.      |
| ENDJOB   | Terminates a job upon completion of FMS statements.                |
| EXPAND   | Concatenates additional DBset members to an existing DBset.        |
| INCLUDE  | Inserts an external file in the input file.                        |
| INIT     | Creates a temporary or permanent DBset.                            |
| NASTRAN  | Specifies values for system cells.                                 |
| PROJ     | Defines the current or default project identifier.                 |

The FMS statements are executed in the following order regardless of their order of appearance in the input file:

NASTRAN, DEFINE  
RFINCLUDE, INCLUDE  
ASSIGN, INIT, EXPAND, DBUPDATE  
PROJECT  
DBCLEAN  
DBFIX  
DBDICT(1)  
DBSETDEL  
ACQUIRE  
RESTART  
DBLOCATE  
DBUNLOAD  
DBLOAD  
DBDIR (2), DBDICT(2)  
ENDJOB

If DBDICT is specified before any of the FMS statements DBSETDEL through DBLOAD, then the directory printout will reflect the processing of DBCLEAN and DBFIX only. If DBDICT is specified after DBSETDEL through DBLOAD, then the directory printout will reflect the processing of all statements in the FMS Section. We recommend that the DBDICT statements be specified last in the FMS Section. Multiple DBLOCATE, DBLOAD, or DBUNLOAD statements are processed in the order in which they appear. If the ENDJOB statement is specified, then only the File Management Section is processed and the Executive Control, Case Control, and Bulk Data Sections are ignored.

## File Management Statement Descriptions

File Management statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

### Description

A brief sentence about the function of the statement is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.



Braces {} indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma as long as the comma is preceded by one or more spaces and no keyword is split across lines. For example:

```
DBLOCATE DATABLK=(KAA) ,
    WHERE (PROJECT='FRONT BUMPER' AND ,
        SEID>0 AND VERSION=4) ,
    LOGI=MASTER3
```

However, if a filename is to be continued on the next line, no space must precede the comma, and the continuation line must have no leading spaces.

### Example

```
ASSIGN      SDB='jw/johannes/Projects/secret/Aero/Tests/wing/,'
Modes/wing_modal.MASTER'
```

Note that all quote marks shown under formats and examples are right-handed single quotation marks and must be entered as such. For example:

```
PROJ='MYJOB'
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks are generally arranged in order of importance and indicate such things as the FMS statement's relationship to other commands, restrictions and recommendations on its use, and further descriptions of the describers.

### WHERE and CONVERT Clauses

The WHERE clause is used in the selection of items (data blocks and parameters) on the DBDICT, DBLOCATE, DBLOAD, and DBUNLOAD statements. The CONVERT clause modifies qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements.

The WHERE and CONVERT clauses specify values for PROJECT, VERSION, qualifiers, and DBSET. PROJECT specifies the project-ID that is originally defined on the PROJECT FMS statement at the time the project is created. VERSION specifies the desired version-ID under the project-ID. Qualifiers are used to uniquely identify items on the database with the same name. For example, data block KAA has SEID as one of its qualifiers, which is the superelement ID. An item may have more than one qualifier and the collection of all qualifiers assigned to an item is called a path. All data blocks and parameters with qualifiers are defined in the NDDL Sequence (NASTRAN Data Definition Language), see [MSC Nastran DMAP Programmer's Guide](#). Data blocks and parameters are defined on the DATABLK and PARAM NDDL statements. The DATABLK and PARAM statements specify the name of the data block, parameter, and also its pathname. The pathnames are defined on the PATH NDDL statement, which lists the qualifiers assigned to the path. Qualifiers are defined on the QUAL NDDL statement. DBSET specifies the desired DBset. The DBset of an item is specified after the LOCATION keyword on the DATABLK and PARAM NDDL statement.

The format of the WHERE clause is:

```
WHERE (where-expr)
```

where-expr is a logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. If the result of the logical expression is TRUE for an item on the database then the item is selected. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2.

A simple where-expr is a comparison using the following relational operators: =, >, <, ≤, ≥, >< or <>. For example, SEID>0 means if SEID is greater than zero, then the logical expression is true. Several simple where expressions may be joined into one where expression by the following logical operators: AND, OR, XOR, and EQV. The NOT operator may be used to negate a where expression. For example, NOT(SEID>0) is the same as SEID≤0. Arithmetic operations and DMAP functions may also be specified in the where expression (see the [MSC Nastran DMAP Programmer's Guide](#).)

If a qualifier in a where-expr is not a qualifier in the path of a specified item, then the where-expr is set to FALSE. If the where-expr does not contain a specification for all qualifiers in the path of an item, then the unspecified qualifiers will be wildcarded (i.e., quali=\*, all values will be selected.) The default values of qualifiers, PROJECT, VERSION, and DBSET are described under the statement in which the WHERE clause is specified.

Examples of the WHERE clause are:

1. Select all items in the database for all superelements except 10 and 30 from Version 1.

```
WHERE (VERSION=1 AND SEID>=0 AND NOT (SEID=10 OR SEID=30))
```

2. Select all entries in database on DBSET=DBALL from all projects and versions.

```
WHERE (PROJECT=PROJECT AND VERSION>0 AND DBSET='DBALL')
```

The CONVERT clause modifies project- and version-ID, DBset-name (see INIT statement), and qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements. It contains one or more assignment statements separated by semicolons. The format of CONVERT clause is:

```
CONVERT (PROJECT=project-expr; VERSION=version-expr; ,
DBSET=DBset-expr; quali=qual-expri[;...])
```

The PROJECT and VERSION statements modify the project-ID (see PROJECT FMS statement) and version-ID. The DBSET statement modifies the DBset-name. The value of quali will be replaced by qual-expri for selected items that have quali in their path. qual-expri is any valid expression (see [Expressions and Operators in the MSC Nastran DMAP Programmer's Guide](#)) containing constants or any qualifier name defined in the path of the item. If qual-expri contains names of qualifiers not in the path of the selected item, then a fatal message is issued. If project-expr and/or version-expr produces a project- or version-ID which does not exist, then one will be created. Also, all version-IDs less than version-expr that do not exist will be created; but they will be “empty.”

Examples of the CONVERT clause are:

1. Set qualifiers SEID, PEID, and SPC to constants 10, 20, 102 respectively.

```
CONVERT (SEID=10;PEID=20;SPC=102)
```

If more than one value of a qualifier is found for an item by the WHERE clause, then each value is processed in qual-expri to define the new qualifier value for each of the selected items. In the following example, if the original values of PEID were 1, 2, and 3; then the new values for the SEID qualifier will be 2, 4, and 6.

2. Set all values of qualifier SEID to be twice the value of the PEID qualifier:

```
CONVERT (SEID=2*PEID)
```

## File Management Statements

### \$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

\$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

**ACQUIRE**

Selects NDDL Schema

Selects the NDDL schema and MSC Nastran delivery database to be used for primary database creation.

Format:

ACQUIRE **NDDL** = { NDDL  
                  nddl-name }

| Descriptor | Meaning   |
|------------|---|
| NDDL       | MSC Nastran NDDL schema.  |
| nddl-name  | Name of a user NDDL schema specified on a COMPILE NDDL statement when the user NDDL was stored. |

Remark:

This statement is used to specify the delivery database when the user wishes to create a solution sequence, yet use the subDMAP objects or NDDL schema or both from the MSC-supplied delivery database.

Example:

The following requests the MSC Nastran NDDL schema to be used when creating a new database.

```
ACQUIRE NDDL
SOL MYDMAP
COMPILE DMAP=MYDMAP, SOUOUT=USROBJ
.
.
.
LINK MYDMAP, SOLOUT=USROBJ
```

**ASSIGN**

Assigns Physical File

Assigns physical file names or other properties to DBset members or special FORTRAN files that are used by other FMS statements or DMAP modules. Also, assigns physical name and/or other properties to modal neutral files (.mnf) for MSC Nastran/ADAMS interface.

**Format 1: Assign a DBset member name**

```
ASSIGN log-name=[*] [filename1]
      [= '*' ] [TEMP] [ DELETE] [ SYS='sys-spec' ]
```

**Format 2: Assign a FORTRAN file**

```
ASSIGN logical-key=[*] [filename2]
      [= '*' ] [UNIT = u]
```

```
[STATUS = ] {NEW  
OLD  
UNKNOWN}
```

```
[FORM = ] {FORMATTED  
UNFORMATTED|UNFORMATTED_64  
UNFORMATTED_32  
UNFORMATTED_MIXED}
```

```
[DEFER ] [TEMP  
DELZERO] [ DELETE] [SYS = 'sys-spec']
```

```
[IMPORT]
```

**Examples:**

1. Assign the DBALL DBset:

```
ASSIGN DB1='filename of member DB1'  
INIT DBALL LOGI=(DB1)
```

2. Assign FORTRAN file 12 to the OUTPUT4 module using the ASCII option:

```
ASSIGN OUTPUT4='filename of FORTRAN file'  
UNIT=12, FORM=FORMATTED
```

3. Assign FORTRAN file to the OPCASE using the ASCII option:

```
ASSIGN OPCASE='filename of FORTRAN file', STATUS=NEW
```

4. Define SYS parameters for the SCR300 DBset file using the default file name:  

```
ASSIGN SCR300 SYS='...'
```
5. Set the default .op2 file format to UNFORMATTED\_64 and assign two .op2 files, one to unit 12 with the file name "test\_op2.12" and one to unit 35 with file name 'test\_op2.35' in ASCII mode.  

```
ASSIGN OUTPUT2 UNFORMATTED_64
...
ASSIGN OUTPUT2='test_op2.12' UNIT=12
ASSIGN OUTPUT2='test_op2.35' UNIT=35 FORM=FORMATTED
```
6. Assign a Fortran unit to the Universal File (UF) containing the FRF information for a test FRF component to be used in an FRF Based Assembly (FBA) process.  

```
ASSIGN UNVFILE='testcomp_unv' UNIT=25
```

| Descriptor  | Meaning   |
|-------------|---|
| log-name    | The name of a DBset member name. log-name may also be referenced on an INIT statement after the LOGICAL keyword.  |
| filename1   | The physical filename assigned to the DBset member. If the default filename (if there is one) is to be used, filename1 may be omitted or specified as * or *. See Remark 6.   |
| logical-key | Specifies defaults for STATUS, UNIT, and FORM of FORTRAN files for other FMS statements, DMAP modules, punching, and plotting operations.   |
| filename2   | The physical file name assigned to the FORTRAN file. If the default filename is to be used, filename2 may be omitted or specified as * or *. See Remark 7.  |
| UNIT=u      | u is the FORTRAN unit number of the FORTRAN file. If this describer is omitted and if filename2 is omitted, this ASSIGN statement will update the defaults for subsequent ASSIGN statements for the same logical-key value. See Remark 7. |
| TEMP        | Requests that the file associated with log-name or logical-key/UNIT be deleted at the end of the run. <b>Warning, using TEMP on OP2 or XDB files will result in the inability to postprocess the results.</b>                             |
| DELETE      | Requests that the file associated with logical-key/UNIT, if it exists before the start of the run, be deleted.  |
| DELZERO     | Requests that the file associated with logical-key/UNIT be deleted at the end of the run if it is zero-length; that is, if it does not contain any data.  |
| STATUS      | Specifies whether the FORTRAN file is being created (STATUS=NEW) or has been created prior to the run (STATUS=OLD). If its status is not known, then STATUS=UNKNOWN is specified.   |
| FORM        | Indicates whether the FORTRAN file is written in ASCII (FORM=FORMATTED) or binary (FORM=UNFORMATTED, UNFORMATTED_64, UNFORMATTED_32, UNFORMATTE_MIX) format. See Remark 10., 11., 12., 13. and 18.  |

| Descriptor | Meaning   |
|------------|---|
| DEFER      | Defers opening/creating the specified file. That is, the file will not be opened/created during MSC Nastran initialization. The file must be explicitly opened by the module or DMAP accessing the file using, for example, FORTIO, before it can be used.  |
| sys-spec   | System-specific, machine-dependent or application-specific controls. For DBset files, these control I/O performance. For FORTRAN files, only the PLOT and DBC Logical-Key Names use this field. For the DBC Logical-Key, these controls are for I/O performance just as for DBset files. For the PLOT Logical-Key, these controls are used for PostScript processing when FORM=FORMATTED is in effect and are ignored otherwise. See Remark 14. |
| RECL = 1   | The size of a block of input/output information specified in words. See Remark 15.  |
| SIZE = s   | The number of blocks allocated to the DBC database. See Remark 16.  |
| IMPORT     | Imports external ifpdb. When this is absent, IFPDB is for export.   |

**Remarks:**

1. The ASSIGN statement and its applications are discussed further in the [Database Concepts in the MSC Nastran Reference Guide](#).
2. The log-name or logical-key descriptor must be the first descriptor on the ASSIGN statement. All other descriptors may appear in any order. With the exception of log-name, logical-key, filename1, filename2, and sys-spec, descriptors and values longer than four characters may be abbreviated to four characters.
3. For FORTRAN files, the logical-key names and their default attributes are listed in [Table 3-1](#). If a logical-key name is identified as “Assignable YES”, then the defaults may be overridden on the ASSIGN statement.
4. Certain reserved names may not be used for log-names or logical-key names. These names are the logical names listed in [Table 3-1](#) that are identified as “Assignable NO”. This list includes: SEMTRN, LNKSWH, MESHFL, LOGFL, INPUT, PRINT, INCLD1, and CNTFL. If they are used, then a User Fatal Message is issued. Unit numbers 1 through 10, 14, 16, 18, 19 and 21 should not be assigned. Up to 4000 ASSIGNs are allowed. Some operating systems may have a smaller limit. To avoid that limit use “ulimit -n 4096”. Unit numbers 1234, 1235, 1236, 1133 to 2269 are not allowed. Some operating systems may have their own limit on the number of open files as well. PUNCH and PLOT may be used, but are not recommended. Most keyword assignments can be specified as command line arguments and/or included in RC files. There are some exceptions such as solve=auto may be specified on the command line or the User RC files, but not in the system RC files (MSC\_BASE/conf/RCfile).
5. If one of the logical-key names indicated in the Remarks 3. and 4. is not specified on this statement, then it is assumed to be a DBset member name log-name as shown in Format 1.
6. If the same log-name is used on more than one DBset ASSIGN statement, the following rules apply:

- a. If there is no current entry for the specified log-name, a new entry in the DBset tables will be created. If there is an existing entry for the specified log-name, the ASSIGN parameters will modify that entry instead of creating a new one.
  - b. If filename1 is omitted or is specified as \* or ‘\*’, the default file name or, if this is a second or subsequent ASSIGN statement for the same log-name, the previously specified file name (or default name if none was previously specified) will be used.
7. If the same logical-key is used on more than one FORTRAN file ASSIGN statement, the following rules apply:
- a. If filename2 is omitted (or specified as \* or ‘\*’) and if the UNIT descriptor is omitted, the ASSIGN parameters will modify the system default entry for the logical-key, establishing the new defaults for any subsequent ASSIGN entry for the logical-key. Note, however, that any entries previously created with the same logical-key will not be modified by the new parameters specified on this ASSIGN statement.
  - b. If the value specified by the UNIT descriptor matches the value for an entry created by a previous ASSIGN statement with a UNIT descriptor, then:
    - If the logical-key values are different, a UFM will be generated; and
    - If the logical-key values are the same, the previous entry will be updated instead of having a new entry created.
  - c. If the value specified by the UNIT descriptor does not match the value for an entry created by a previous ASSIGN statement with a UNIT descriptor, then a new entry will be created in the FORTRAN unit tables.
  - d. If the file name is omitted or specified as \* or ‘\*’, the default file name or, if this is a second or subsequent ASSIGN statement for the same logical-key/UNIT combination, a previously specified file name (or default name if none was previously specified) will be used.
8. If you are using IFPSTAR (default), you can use below command to import IFPDAT file created by the initial run if the file is not located in the same directory as MASTER and DBALL files. ASSIGN IFPDDB=first\_run\_directory/run1.IFPDAT IMPORT.
9. STATUS, UNIT, and FORM are ignored if assigning a log-name (DBset member name).
10. FORM=FORMATTED must be specified for a unit when:
- ASCII output is desired from the OUTPUT4 DMAP modules that processes the unit. See the [MSC Nastran DMAP Programmer's Guide](#).
  - FORMAT=NEUTRAL is selected on the DBUNLOAD and DBLOAD FMS statements that process the unit. See the [Database Concepts in the MSC Nastran Reference Guide](#).
  - The neutral file format is desired for the OUTPUT2 module.
  - PostScript output is desired for PLOT requests when “PLOTTER NAST” is in effect.

11. For the DBUNLOAD FMS statement and the OUTPUT2 and OUTPUT4 modules, binary format may be requested using FORM=UNFORMATTED, FORM=UNFORMATTED\_64, FORM=UNFORMATTED\_32 and FORM=UNFORMATTED\_MIX. The output formats with these keywords are as the following:

| FORM            | Table (bit) | Matrix (bit) | Old keyword    |
|-----------------|-------------|--------------|----------------|
| UNFORMATTED     | 64          | 64           | UNFORMATTED    |
| UNFORMATTED_64  | 64          | 64           | LITTLEENDIAN64 |
| UNFORMATTED_32  | 32          | 32           | N/A            |
| UNFORMATTED_MIX | 32          | 64           | LITTLEENDIAN   |

12. The FORM= descriptor is ignored for the DBLOAD FMS statement and INPUTT2 and INPUTT4 modules. MSC Nastran determines the actual file format when it accesses the specified file. If the FORM= descriptor is specified on an ASSIGN statement for these logical-keys, the syntax of the descriptor will be validated but will otherwise be ignored. However,
- For non-native binary files the INPUTT2 modules can only process data blocks with an NDDL description. (See the [MSC Nastran DMAP Programmer's Guide](#) under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be processed with FORM=UNFORMATTED if TYPE=UNSTRUCTURED, KDICT, or KELM.
  - Although formatted files are machine independent, if the file in unformatted DBLOAD can only process input files in native binary format.
13. For the DBUNLOAD FMS statement and OUTPUT2 module, if FORM is other than UNFORMATTED (or equivalent, e.g., UNFORMATTED\_32 on a Linux or Windows platform), then only data blocks with an NDDL description are processed. (See the [MSC Nastran DMAP Programmer's Guide](#) under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be processed with FORM=UNFORMATTED if TYPE=UNSTRUCTURED, KDICT, or KELM.
14. See the [MSC Nastran 2020 Installation and Operations Guide](#) for further information on sys-spec controls and on machine-dependent aspects of the ASSIGN statement. Also, if there are SYS specifications on more than one ASSIGN statement specifying the same log-name or logical-key/UNIT combination, the second and subsequent specifications will appended to the current SYS specification with a comma separator.
15. Currently the RECL keyword is used by the DBC module and has a default minimum of 1024 words. The maximum allowed is 65536 words and is used to increase the database capacity.
16. The SIZE keyword is used by the DBC module and has a default of 16777215. The maximum allowed is 2147483647 and is used to increase the database capacity. Patran releases before 2001 should use the defaults for RECL and SIZE or database verification failures will occur.
17. logical-key name MNF does not utilize UNIT or FORM.

18. For logical-key DBC, if the .xdb file is new, the desired binary format may be specified in the same way as for the OUTPUT2 and OUTPUT4 modules, as described in Remark 11., except that FORM=FORMATTED is not valid. If the .xdb file is not new, the FORM= descriptor is ignored and MSC Nastran determines the format of the existing .xdb file. MSC Nastran can read and or update an .xdb file in any valid format. Note, on long-word systems (mode=i8) the effect of 64-bit output files is that ADAMS(MNF), Patran, and SimXpert may fail to process the results.
19. SOL700 reserves unit numbers 87-92 for its internal use. When using SOL700 users should not use these unit number to avoid conflicts.
20. The total length of any line in an ASSIGN statement must not exceed 72 characters. Long file names may be split across multiple lines with commas at the end of line and next to be without spaces. For example, the file: *ASSIGN SCR300='E:\hrishikesh\_scr300\testing\delimiter\issue\scratch.SCR300' DELETE* should be specified with the following input: *ASSIGN SCR300='E:\hrishikesh\_scr300\testing\delimiter\issue\scratch.SCR300' DELETE*.

**Table 3-1 FORTRAN Files and Their Default Attributes**

| Logical Key Name | Physical Name | Unit No. | Form          | Status | Assignable | Open | Access | Description/Application   |
|------------------|---------------|----------|---------------|--------|------------|------|--------|---------------------------|
| SEMTRN           | sdir/data.f01 | 1        | FORMATTED     | NEW    | NO         | YES  | SEQ.   | Input Data Copy Unit      |
| LNKSWH           | sdir/data.f02 | 2        | UNFORMATTED   | NEW    | NO         | YES  | SEQ.   | Link Switch Unit          |
| MESHFL           | sdir/data.f03 | 3        | FORMATTED     | NEW    | NO         | YES  | SEQ.   | Input Data Copy Unit      |
| LOGFI            | out.f04       | 4        | FORMATTED     | NEW    | NO         | YES  | SEQ.   | Execution Summary Unit    |
| INPUT            | data.dat      | 5        | FORMATTED     | OLD    | NO         | YES  | SEQ.   | Input File Unit           |
| PRINT            | out.f06       | 6        | FORMATTED     | NEW    | NO         | YES  | SEQ.   | Main Print Output Unit    |
| PUNCH            | out.pch       | 7        | FORMATTED     | NEW    | YES        | YES  | SEQ.   | Default Punch Output Unit |
|                  | authorize.dat | 8        | FORMATTED     | OLD    | NO         | YES  | SEQ.   | Authorization File        |
| INCLD1           |               |          |               |        | NO         |      |        | Available for Use         |
| CNTFL            |               |          |               |        | NO         |      |        | Available for Use         |
| INPUTT2          | REQ           | REQ      |               | OLD    | YES        | NO   | SEQ.   | INPUTT2 Unit              |
| OUTPUT2+         | out.op2       | 12       | UNFORMATTED*  | NEW    | YES        | YES  | SEQ.   | OUTPUT2 Unit              |
| INPUTT4          | REQ           | REQ      |               | OLD    | YES        | NO   | SEQ.   | INPUTT4 Unit              |
| OUTPUT4          | REQ           | REQ      | UNFORMATTED*  | NEW    | YES        | NO   | SEQ.   | OUTPUT4 Unit              |
| PLOT             | out.plt       | 14       | UNFORMATTED++ | NEW    | YES        | YES  | SEQ.   | Plotter Output Unit       |
| BULKECHO         | out.becho     | 18       | FORMATTED     | NEW    | YES        | YES  | SEQ.   | Bulk EchoUnit             |
| OUTPUT2F         | out           | 19       | UNFORMATTED   | NEW    | YES        |      | SEQ.   | Named OUTPUT2 Pattern     |
| OPCASE           | REQ           | 22       | FORMATED      | NEW    | YES        |      | SEQ.   | Available for Use         |
| TOPDES           | out.des       | 21       | FORMATTED     | NEW    | YES        | YES  | SEQ.   | Topology Optimization     |
| AESO             | out.AESO      | 23       | FORMATTED     | NEW    | YES        | YES  | SEQ.   | Optimization              |
| DBC              | out.xdb       | 40       | UNFORMATTED   | NEW    | YES        | YES  | DIRECT | Database Converter Unit   |
| DBUNLOAD         | REQ           | 50       | UNFORMATTED*  | NEW    | YES        | NO   | SEQ.   | DBUNLOAD FMS statement    |

Table 3-1 FORTRAN Files and Their Default Attributes (continued)

| Logical Key Name | Physical Name     | Unit No. | Form      | Status | Assignable | Open | Access | Description/Application   |
|------------------|-------------------|----------|-----------|--------|------------|------|--------|---|
| DBLOAD           | REQ               | 51       |           | OLD    | YES        | NO   | SEQ.   | DBLOAD FMS statement  |
| MNF              | <i>out.mnf</i>    | none     | none      | NEW    | YES        | NO   | SEQ.   | Interface for ADAMS/Flex  |
| A502LU           |                   |          |           |        |            |      |        | Available for Use   |
| DBMIG            |                   |          |           |        |            |      |        | Available for Use   |
| USERFILE         | REQ               | REQ      | REQ       | REQ    | YES        | NO   | SEQ.   | Any User-Defined File   |
| UNVFILE          | REQ               | REQ      | FORMATTED | OLD    | YES        | NO   | SEQ.   | Universal File (UF) Unit  |
| ADFFILE          | <i>out.afu</i>    | none     | none      | NEW    | YES        | NO   | SEQ.   | Associated Data File Unit   |
| IFPDB            | <i>out.IFPDB</i>  | no       | no        | NEW    | YES        | NO   | SEQ    | Export or import IFPDAT file  |
| ACG              | <i>out.acg</i>    | no       | no        | NEW    | YES        | NO   | SEQ    | Automatic Contact Generation file   |
| HDF5             | <i>out.h5</i>     | no       | no        | NEW    | YES        | NO   | N/A    | Nastran HDF5 result database  |
| HDF5IN           | <i>pre_run.h5</i> | REQ      | no        | OLD    | YES        | NO   | SEQ    | Read displacement or eigenvector datasets from MSC Nastran hdf5 result database |
| IMPFIN           | example.impf      | REQ      | no        | OLD    | YES        | NO   | N/A    | Read imperfection file  |

where:

|                  |   |
|------------------|---|
| Logical Key Name | Specifies the logical-key NAME used on the ASSIGN statement.  |
| Physical Name    | Specifies the default name used to open the file; i.e., the default filename2 name.<br>“REQ” means that this parameter is required in the ASSIGN statement from the user. |
| Unit No.         | Specifies the default FORTRAN unit number used by MSC Nastran.<br>“REQ” means that this parameter is required in the ASSIGN statement from the user.                      |
| Form             | Specifies the default FORM used when the file is opened.  |
| Status           | Specifies the default STATUS used when the file is opened.<br>“REQ” means that this parameter is required in the ASSIGN statement from the user.                          |
| Assignable       | If “YES”, the user may assign a physical file to this logical name.<br>If “NO”, the unit (if any) and logical name are reserved by MSC Nastran.                           |
| Open             | If “YES”, the file is opened by default.<br>If “NO”, the file must be explicitly opened.  |

|        |   |
|--------|---|
| Access | If “SEQ”, the file is opened for sequential access.<br>If “DIRECT”, the file is opened for direct access. |
| sdir   | The scratch directory specified using the “sdirectory” keyword.   |
| data   | The name of the input data file with all directory and extensions removed.                                |
| out    | The directory and file prefix specified using the “out” keyword or taken by default.                      |

- Notes:**
- + The actual logical-key name for this is “.op2”. If you use “OUTPUT2” (even though this is still the logical-key name put out by Patran) you will get a User Fatal Message from MSC Nastran.
  - \* FORMATTED is required for neutral-format OUTPUT2 files and ASCII-format OUTPUT4 files.
  - ++ If FORM=FORMATTED is specified, the default extension is changed to “.ps” and, if PLOTTER NAST is requested (PLOTTER SC is not supported in PostScript mode), the plot data will be generated in PostScript format directly instead of having to use the PLOTPS or MSCPLOTPS utility programs. Any desired plotting options may be specified as keywords in the SYS= field of the ASSIGN statement or using the SYSFIELD command-line option, specifying PLOT (keyword=value, ...). The valid keywords are the same as those that can be specified for the PLOTPS or MSCPLOTPS utility programs except that the “begin”, “debug”, “dump”, “end”, “format” and “output” keywords are not allowed. PostScript mode can be made the default by specifying the following ASSIGN statement in an RC file:

```
ASSIGN PLOT=*, FORM=FORMATTED
```

This ASSIGN statement may also include a SYS= specification if any special PostScript keywords are to be used for every plot.

## CONNECT

### User Defined Service and Group Evaluator Data

Defines the newly introduced Simulation Component Architecture (SCA) User Defined Service (UDS) into MSC Nastran via a new CONNECT SERVICE statement. Bulk Data entries such as NLRSFD and MATUDS refer to this new service.

External geometric or beam cross section entities, external design responses, and external splines are still defined through the old grouping also described below. These entities should belong to the same evaluator-class (set of routines that process them), and in the case of geometric data, should reside on the same database.

1. A group of external geometric entities. These entities should belong to the same evaluator-class (a set of routines that process them), and should reside on the same database. The GMCURV and GMSURF Bulk Data entries relate to the GEOMEVAL type.
2. A group of external beam cross section entities. These entities should belong to the same evaluator-class (a set of routines that process them). The PBARL and PBEAML entries relate to the BEAMEVAL type.
3. A group of external spline entities. The SPLINEX Bulk Data entry relates to the SPLINEX type.
4. A group of external design response entities. The DRESP3 Bulk Data entry relates to the DRESP3 type.

**UDS Format:**

CONNECT SERVICE <service\_identifier> <service\_name>

**Old Group Format:**

|         |  |  |
|---------|--|--|
| CONNECT | <div style="display: inline-block; vertical-align: middle; border-left: 1px solid black; padding-left: 10px; margin-right: 10px;"> <b>GEOMEVAL</b><br/> <b>BEAMEVAL</b><br/> <b>DRESP3</b><br/> <b>SPLINEX</b> </div> <span style="font-size: 2em;">[</span> | <span style="font-size: 2em;">]</span> group evaluator 'path' 'data' |
|---------|--|--|

**UDS Examples:**

CONNECT SERVICE mysub 'SCA.MDSolver.Util.Ums'

Creates a service identifier "mysub" which points to the "SCA.MDSolver.Util.Ums" service.

CONNECT SERVICE MYSUB 'ExtServ.Nlrsfd'

In this case, the user is requesting the NLRSFD Bulk Data entry with the GRPNAME MYSUB obtain its characteristics from the externally connected service defined in the CDL file under the component keyword. The user must create an external shared object library (dynamic link library on Windows) using the SCA Scons utility and configure it to function with the MSC Nastran executable.

**Old Group Examples:**

CONNECT GEOMEVAL FENDER, CATIA, '/u/kiz/adp', 'Version=6 as of 1/31/93'

In this case, the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as FENDER use the CATIA database/evaluator. For each GMCURV and GMSURF entry where the group parameter is set to FENDER, appropriate evaluator routines will be called to initialize and perform computations on the curve or surface.

**CONNECT GEOMEVAL HOOD, MSCRPC**

In this case, the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as HOOD use the Nastran RPC database/evaluator. There is no need for additional routines to be supplied by the user since the MSCRPC and MSCEQN evaluator libraries are included in the standard Nastran delivery.

**CONNECT GEOMEVAL DOOR, MSCEQN**

In this case, the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as DOOR use the Nastran EQUATION database/evaluator. There is no need for additional routines to be supplied by the user since the MSCRPC and MSCEQN evaluator libraries are included in the Nastran standard delivery.

**CONNECT BEAMEVAL HOIST, NEWBEAMS**

In this case, the user is requesting that all calculations on PBARL and PBEAML Bulk Data entries that are grouped as HOIST use the NEWBEAMS evaluator. In this case, the user must supply the NEWBEAMS beam cross section evaluator library, and configure it to function with the Nastran executable program.

**CONNECT DRESP3 TAILWING, EXTRESP**

In this case, the user is requesting that all calculations on DRESP3 Bulk Data entries, that are grouped as TAILWING use the EXTRESP evaluator. Thus, the user must create the EXTRESP external response server program, and configure it to function with the MSC Nastran executable.

**CONNECT SPLINEX SPLNGRP EXTSPLN**

In this case, the user is requesting that all calculations on SPLINEX Bulk Data entries, that are grouped as SPLNGRP, use the EXTPNL evaluator. Thus, the user must create the EXTSPLN external spline server program and configure it to function with the Nastran executable.

| Descriptor<br>(UDS Format) | Meaning   |
|----------------------------|---|
| group                      | Group name referenced by the GROUP field on the NLRSFD and MATUDS Bulk Data entries.  |
| service_identifier         | service_identifier is a name tag (8 characters long) which will be used to identify the service implementation to be used for a specific bulk data entry. |
| service_name               | service_name is the name of service   |

| Descriptor<br>(Old Group Format) | Meaning   |
|----------------------------------|---|
| group                            | Group name referenced by the GROUP field on DRESP3, GMCURV, GMSURF, PBTRL, PBEAML, and SPLINEX Bulk Data entries.   |
| evaluator                        | Identifies the particular class of evaluator to which the geometric, beam cross section, external response, or external spline entities belong. Entities belonging to one evaluator-class are handled by the same set of routines (either MSC-provided or user-provided). For geometry, two classes of evaluators are provided internally with MSC Nastran. They are MSCRPC (rational parametric cubic) and MSCEQN (generic equation). For beam cross sections, the class MSCBML (MSC Beam Library) is provided internally. Users may develop custom evaluator libraries for geometry, beam cross sections, external responses, or external splines and configure them for use with MSC Nastran. See Remarks 4., 5., and 7. |
| path                             | Optional pathname or filename used by evaluator. Path must be enclosed by single quotation marks if it contains lowercase characters.   |
| data                             | Optional character string passed to the evaluator. Data must be enclosed by single quotation marks if it contains lowercase characters or embedded blanks.  |

#### Remarks (UDS Format):

1. The process of enabling User Defined Services in MSC Nastran consists of the following four steps:
  - a. Creating the desired implementation for the User Defined Services in the form of dynamic-link libraries,
  - b. Defining the location of the user defined service, service catalogue, service resource directory,
  - c. Specifying the proper commands in the model to load the service,
  - d. Identifying the elements that use the user supplied implementation.
2. In order to create a dynamic-link library suitable for usage with MSC Nastran, a build environment is delivered to assist the user in building the library. Please refer to the *Simulation Component Architecture Guide* and the *User Defined Service Guide* which describe the features and capabilities of the build system.

#### Remarks (Old Group Format):

1. CONNECT requests:
  - An external data base or evaluator, or
  - A user-defined grouping for geometric data defined by GMCURV and GMSURF entries, or beam cross section data defined by PBTRL and PBEAML entries.

2. Two reserved group names, MSCGRP0 and MSCGRP1, have been predefined for geometric entities. These names may be used in the GMCURV and GMSURF entries without being defined explicitly by means of a CONNECT FMS statement. The group MSCGRP0 corresponds to the MSCRPC (rational parametric cubic) evaluator and the group MSCGRP1 corresponds to the MSCEQN (Generic Equation) evaluator.
3. A single reserved group name, MSCBML0, has been predefined for beam cross section entities. It may be used in the PBARL and PBEAML entries without being defined explicitly by means of a CONNECT FMS statement. It corresponds to the MSCBML (MSC Beam-Library) evaluator.
4. Custom geometric evaluator libraries developed by users should comply with the *MSC Nastran Geometry Evaluator Developer's Guide*.
5. Custom beam cross section evaluator libraries developed by users should comply with the guidelines in the *MSC Nastran V69 Release Guide*, Section 3.1, Beam Cross-Section Library, and Appendix C: Adding Your Own Beam Cross-Section Library.
6. Custom responses developed by users should comply with the procedures and guidelines in "Support of External Response in SOL 200" on page 55 of the *MSC Nastran 2004 Release Guide*.
7. Once developed, an evaluator may be configured as:
  - Internal, where the evaluator routines are linked with the rest of the Nastran object modules to comprise the Nastran executable program; or
  - External, where the evaluator routines are linked with an MSC-provided server program to constitute an independent geometry server.

**DBCLEAN****Deletes Database Versions and/or Projects**

Deletes one or more versions and/or projects from the database.

**Format:**

DBCLEAN VERSION = {version-ID,\*} [PROJECT={project-ID,\*}]

| Descriptor | Meaning   |
|------------|---|
| version-ID | Version identifier of the database to be deleted.   |
| *          | Wildcard. All versions or projects to be deleted.   |
| project-ID | Project identifier of the project to be deleted. (See the FMS statement, <a href="#">PROJ, 103</a> .) |

**Remarks:**

1. There may be up to ten DBCLEAN statements in the FMS Section.
2. If no project-ID is given, the current project-ID is assumed.

**Example:**

DBCLEAN VERS = 7 PROJ = 'OUTER WING - LEFT'

The preceding example would delete from the database all data blocks and parameters stored under Version 7 of the project identified as OUTER WING - LEFT.

**DBDICT**

Prints Database Directory Tables

DBDICT prints the following database directory tables:

- Data blocks described by an NDDL DATABLK statement.
- Parameters described by an NDDL PARAM statement.
- All unique paths (KEYs) and their qualifier values.
- Qualifiers and their current values.
- Data blocks not described by an NDDL DATABLK statement.
- Parameters not described by an NDDL PARAM statement.
- Project and version information.

#### Basic Format:

The basic format of DBDICT specifies which tables to print and prints all items (data blocks and parameters) found in the directory. Also, the attributes (colnames) to be printed, and the print format, are predefined. Note that more than one table may be specified on the same DBDICT statement.

```
DBDICT [DATABLK PARAM PROJVERS QUALCURR QUALIFIERS]
```

#### Examples:

```
DBDICT
```

```
DBDICT PARAM PROJVERS
```

#### Full Format:

The full format permits the selection of items by name and/or by the WHERE descriptor. The full format also permits the attributes to be printed using the SELECT descriptor. In addition, the print format can be specified with the SORT, FORMAT, and LABEL descriptors. Note that the full format only allows the specification of a single table on a DBDICT statement.

|        |  |                    |
|--------|--|--------------------|
| DBDICT | $\left[ \begin{array}{l} \text{DATABLK} \\ \text{DATABLK(LOCAL)} \end{array} \right] = \left[ \begin{array}{l} * \\ (\text{datblk-list}) \end{array} \right]$ $\left[ \begin{array}{l} \text{PARAM} \\ \text{PARAM(LOCAL)} \end{array} \right] = \left[ \begin{array}{l} * \\ (\text{param-list}) \end{array} \right]$ $\left[ \begin{array}{l} \text{PROJVERS} \\ \text{QUALCURR} \\ \text{QUALIFIERS} \end{array} \right]$ | WHERE(where-expr), |
|--------|--|--------------------|

```
SELECT(colname[- ' col-label']...),
```

```
FORMAT      (FWIDTH = w [.d] DWIDTH = w [.d] AWIDTH = a IWIDTH = i,
             LWIDTH = k COLSPACE = c VALUE = w,
             colname = col-width, . . .),
```

SORT $\left( \text{colname} \left[ = \begin{bmatrix} \text{A} \\ \text{D} \end{bmatrix}, \dots \right] \right)$ ,

LABEL $\left( \text{'page - title'} \left[ \begin{array}{c} \text{RIGHT} \\ \text{CENTER} \\ \text{LEFT} \end{array} \right] \right)$

| Descriptor | Meaning  |
|------------|--|
| DATABLK    | Print the data blocks. datablk-list specifies a list of NDDL-defined data blocks separated by commas. If LOCAL is specified, the non-NDDL-defined data blocks are printed.   |
| PARAM      | Print the parameter table. param-list specifies a list of parameters separated by commas. If LOCAL is specified, the non-NDDL-defined parameters are printed.  |
| PROJVERS   | Print the project-version table.   |
| QUALIFIERS | Print the qualifier table.   |
| QUALCURR   | Print the current values of the qualifiers. SORT is ignored.   |
| where-expr | Logical expression that specifies the desired values of colnames described below. For example, WHERE(VERSION=4 AND SEID < > 2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for a further description. The default for VERSION is the last version, and PROJECT is the current project. The default for qual is*, which is all qualifier values found on the database. See also Remark 12. |
| SELECT     | Specifies a list of column names to be printed. The order of the specified colnames will be printed from left to right. If colname is not specified, then all columns will be printed.   |
| colname    | Column name. Colname specifies a particular attribute of the database item such as data block name (NAME), creation date (CDATE), number of blocks (SIZE), or qualifier name (SEID, SPC, etc.). The allowable colnames are given in the Remarks.   |
| col-label  | The label to printed above the column identified by colname. The default for col-label is the colname. col-label may not be specified for the following colnames: QUALSET, QUALALL, and TRAILER.   |
| FWIDTH=w.d | Specifies the default width for single-precision real numbers in real and complex qualifiers (Integers: w>0 and d>0; Default=12.5).  |
| DWIDTH=w.d | Specifies the default width for double-precision real numbers in real and complex qualifiers (Integers: w>0 and d>0; Default=17.10).   |
| AWIDTH=a   | Specifies the default width for character string qualifiers. Character strings are printed with enclosing single quotation marks, even if the string is blank (Integer>0; Default=8).  |

| Descriptor             | Meaning  |
|------------------------|--|
| IWIDTH=i               | Specifies the default width for integer qualifiers (Integer>0; see Remarks for defaults).  |
| LWIDTH=k               | Specifies the default width for logical qualifiers. Logical values are printed as either "T" for TRUE or "F" for FALSE (Integer>0; Default=1).   |
| COLSPACE=c             | Specifies the default number of spaces between columns (Integer>0; see Remarks for defaults).  |
| VALUE=w                | Specifies the default width for parameter values. The values are printed as character strings with left justification (Integer>0; Default=40)  |
| col-width              | The print width of the data under colname or qual-name. For real numbers, specify w.d where w is the width of the field and d is the number of digits in the mantissa. For integers and character strings, specify w, where w is the width of the field. col-width may not be specified for colnames QUALSET, QUALALL, and TRAILER.  |
| SORT                   | Specifies how the rows are sorted. The sort is performed in order according to each colname specified in the list. A "D" following the colname causes the sort to be in descending order. An "A" following the colname causes the sort to be in ascending order. Colnames QUALSET, QUALALL, and TRAILER may not be specified under SORT. Each colname specified in SORT must be separated by commas. |
| page-title             | A title to be printed on each page of the directory output.  |
| RIGHT,<br>CENTER, LEFT | Print justification of the page title.   |

**Remarks:**

1. DBDICT prints seven different tables according to a default or a user-defined format. The tables are:

Table 1 DBDICT Tables

| Descriptor | Description  | Default Page-Title | See Remark |
|------------|--|--------------------|------------|
| DATABLK    | Data blocks described by a NDDL DATABLK statement. | NDDL DATABASES     | 2.         |
| PARAM      | Parameters described by a NDDL PARAM statement.    | NDDL PARAMETERS    | 3.         |
| QUALCURR   | Current qualifiers and their values.               | CURRENT QUALIFIERS | 4.         |

Table 1 DBDICT Tables (continued)

| Descriptor     | Description  | Default Page-Title | See Remark         |
|----------------|--|--------------------|--------------------|
| QUALIFIERS     | Qualifiers and their values for each key number.       | QUALIFIERS         | <a href="#">5.</a> |
| DATABLK(LOCAL) | Data blocks not described by a NDDL DATABLK statement. | LOCAL DATABLOCKS   | <a href="#">6.</a> |
| PARAM(LOCAL)   | Parameters not described by a NDDL PARAM statement.    | LOCAL PARAMETERS   | <a href="#">7.</a> |
| PROJVERS       | Project-Version.                                       | PROJECT-VERSION    | <a href="#">8.</a> |

If DBDICT is specified without any descriptors, then the NDDL Data blocks Table will be printed. See Remark [2](#).

DATABLK(LOCAL) and PARAM(LOCAL) produce no output, and QUALCURR produces the default values specified on the NDDL QUAL statement.

The defaults and allowable colnames for SELECT, FORMAT, SORT, and LABEL depend on the table. The defaults are described in the following remarks and tables.

2. The default print of the NDDL Data Blocks Table is obtained by

DBDICT

or

DBDICT DATABLK

and is equivalent to

```
DBDICT DATABLK ,
  SELECT (NAME, DATABASE, DBSET, PROJ, VERS, CDATE, CTIME,
    SIZE, KEY, PURGED='PU', EQUIVD='EQ',
    POINTER='FILE', QUALSET) ,
  FORMAT (NAME=8, DBSET=8, CDATE=6, CTIME=6, SIZE=5,
    KEY=4, PURGED=4, EQUIVD=4, POINTER=8,
    IWIDTH=5, COLSPACE=1) ,
  SORT (PROJ=A, VERS=A, DBSET=A, NAME=A) ,
  LABEL ('NDDL DATABLOCKS' CENTER)
```

and looks like:

| * * * * D I C T I O N A R Y P R I N T * * * *   |          |       |      |      |        |       |                |            |      |     |        |    |      |      |      |      |     |     |      |
|---|----------|-------|------|------|--------|-------|----------------|------------|------|-----|--------|----|------|------|------|------|-----|-----|------|
| EXECUTION OF DMAP STATEMENT NUMBER 20<br>MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER 16 |          |       |      |      |        |       |                |            |      |     |        |    |      |      |      |      |     |     |      |
| NAME  | DATABASE | DBSET | PROJ | VERS | CDATE  | CTIME | NDL DATABLOCKS |            |      |     |        |    |      |      |      |      | SPC | MPC | METH |
|   |          |       |      |      |        |       | NDL            | DATABLOCKS | SIZE | KEY | PU     | EQ | FILE | SEID | PEID | LOAD |     |     |      |
| AGG   | MASTER   | DBALL | 1    | 1    | 930805 | 72340 | 0              | 326        | 1    | 0   | 132484 | 0  | 0    |      |      |      |     |     |      |
| AXIC  | MASTER   | DBALL | 1    | 1    | 930805 | 72336 | 0              | 315        | 1    | 0   | 65764  |    |      |      |      |      |     |     |      |
| BGPDTS  | MASTER   | DBALL | 1    | 1    | 930805 | 72338 | 1              | 324        | 0    | 2   | 131332 |    |      |      |      |      |     |     |      |
| BGPDTX  | MASTER   | DBALL | 1    | 1    | 930805 | 72338 | 1              | 324        | 0    | 1   | 131332 |    |      |      |      |      |     |     |      |
| BJJ   | MASTER   | DBALL | 1    | 1    | 930805 | 72341 | 0              | 332        | 1    | 0   | 132612 |    |      |      |      |      |     |     |      |
| BULK  | MASTER   | DBALL | 1    | 1    | 930805 | 72336 | 2              | 315        | 0    | 0   | 65700  |    |      |      |      |      |     |     |      |
| CASECC  | MASTER   | DBALL | 1    | 1    | 930805 | 72336 | 1              | 316        | 0    | 2   | 67428  |    |      |      |      |      |     |     |      |

Figure 3-1 DBDICT DATABLK Example

Table 2 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 2 DBDICT DATABLK Colnames

| colname   | Default col-width | Default col-label | Description                                |
|-----------|-------------------|-------------------|--|
| PROJECT   | 40                | PROJECT NAME      | Project name defined by PROJECT statement. |
| PROJ      | 4                 | PROJ NO           | Project number associated with PROJECT.    |
| VERS      | 4                 | VERSION           | Version number.                            |
| CDATE     | 6                 | CDATE             | Creation date.                             |
| CTIME     | 6                 | CTIME             | Creation time.                             |
| NAME      | 8                 | NAME              | Parameter name.                            |
| DATABASE  | 8                 | DATABASE          | MASTER DBset name.                         |
| DBSET     | 8                 | DBSET             | DBset name.                                |
| RDATE     | 6                 | RDATE             | Revision date.                             |
| RTIME     | 6                 | RTIME             | Revision time.                             |
| SIZE      | 5                 | SIZE              | Number of blocks.                          |
| qual-name | See Note.         | qualifier name    | Qualifier name.                            |
| KEY       | 4                 | KEY               | Key number.                                |
| TRLi      | 8                 | TRLi              | i-th word in the trailer.                  |
| TRAILER   | 8                 | TRLi              | All 10 trailer words.                      |
| EXTNAME   | 8                 | EXTNAME           | Extended name.                             |
| EQUIVD    | 4                 | EQ                | Equivalenced flag.                         |
| PURGED    | 4                 | PU                | Purged flag.                               |

Table 2 DBDICT DATABLK Colnames (continued)

| colname  | Default col-width | Default col-label | Description                          |
|----------|-------------------|-------------------|--------------------------------------|
| EQFLAG   | 4                 | EF                | Scratch equivalenced flag.           |
| SCRFLAG  | 4                 | SF                | Scratch DBSET flag.                  |
| POINTER  | 8                 | POINTER           | Directory pointer.                   |
| DBENTRY  | 8                 | DBENTRY           | Database entry pointer.              |
| FEQCHAIN | 8                 | FEQCHAIN          | Forward equivalence chain.           |
| BEQCHAIN | 8                 | BEQCHAIN          | Backward equivalence chain.          |
| DBDIR20  | 9                 | DBDIR(20)         | Directory word 20.                   |
| QUALALL  | See Note.         | qualifier name    | All qualifiers.                      |
| QUALSET  | See Note.         | qualifier name    | Predefined subset of all qualifiers. |

**Note:**

Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, AWIDTH=8, and FWIDTH=12.5.

3. The default print of the NDDL Parameter Table is obtained by

DBDICT PARAM

and is equivalent to

```
DBDICT PARAM,
  SELECT (NAME, DATABASE, DBSET, PROJ, VERS, CDATE, CTIME,
          KEY, VALUE, QUALSET),
          FORMAT (NAME=8, DATABASE=8, DBSET=8, CDATE=6, CTIME=6,
                  KEY=4, VALUE=40, IWIDTH=5, COLSPACE=1),
          SORT (PROJ=A, VERS=A, DBSET=A, NAME=A),
          LABEL ('NDDL PARAMETERS' CENTER)
```

and looks like:

| * * * * * D I C T I O N A R Y P R I N T * * * * *   |          |        |      |      |        |       |                |              |      |      |      |     |     |      |
|---|----------|--------|------|------|--------|-------|----------------|--------------|------|------|------|-----|-----|------|
| EXECUTION OF DMAP STATEMENT NUMBER 21<br>MODULE NAME = DBDICT , SUBDMAF SEKRRS , OSCAR RECORD NUMBER 17 |          |        |      |      |        |       |                |              |      |      |      |     |     |      |
| NAME  | DATABASE | DBSET  | PROJ | VERS | CDATE  | CTIME | NDL PARAMETERS |              | SEID | PEID | LOAD | SPC | MPC | METH |
|   |          |        |      |      |        |       | KEY            | VALUE        |      |      |      |     |     |      |
| ACOUSTIC  | MASTER   | MASTER | 1    | 1    | 930805 | 72338 | 323            | 0            | 0    | 0    | 0    |     |     |      |
| ALTRRED   | MASTER   | MASTER | 1    | 1    | 930805 | 72338 | 319            | NO           |      |      |      |     |     |      |
| BCHNG   | MASTER   | MASTER | 1    | 1    | 930805 | 72337 | 325            | FALSE        |      |      |      |     |     |      |
| DBALLX  | MASTER   | MASTER | 1    | 1    | 930805 | 72336 | 318            | DBALL        | -1   | -1   |      |     |     |      |
| EPSBIG  | MASTER   | MASTER | 1    | 1    | 930805 | 72339 | 323            | 1.000000E+12 | 0    | 0    |      |     |     |      |
| ERROR   | MASTER   | MASTER | 1    | 1    | 930805 | 72338 | 319            | -1           |      |      |      |     |     |      |
| FIXEDB  | MASTER   | MASTER | 1    | 1    | 930805 | 72338 | 323            | 0            | 0    | 0    | 0    |     |     |      |

Figure 3-2 DBDICT PARAM Example

Table 3 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 3 DBDICT PARAM Colnames

| colname   | Default col-width | Default col-label | Description                                |
|-----------|-------------------|-------------------|--|
| PROJECT   | 40                | PROJECT NAME      | Project name defined by PROJECT statement. |
| PROJ      | 5                 | PROJ              | Project number associated with PROJECT.    |
| VERS      | 4                 | VERS              | Version number.                            |
| CDATE     | 6                 | CDATE             | Creation date.                             |
| CTIME     | 6                 | CTIME             | Creation time.                             |
| NAME      | 8                 | NAME              | Parameter name.                            |
| DATABASE  | 8                 | DATABASE          | MASTER DBset name.                         |
| DBSET     | 8                 | DBSET             | DBset name.                                |
| RDATE     | 6                 | RDATE             | Revision date.                             |
| RTIME     | 6                 | RTIME             | Revision time.                             |
| POINTER   | 8                 | POINTER           | Directory pointer.                         |
| VALUE     | 40                | VALUE             | Parameter value.                           |
| KEY       | 4                 | KEY               | Key number.                                |
| qual-name | See Note.         | qualifier name    | Qualifier name.                            |
| QUALALL   | See Note.         | qualifier name    | All qualifiers.                            |
| QUALSET   | See Note.         | qualifier name    | Predefined subset of all qualifiers.       |

**Note:** Default widths for qualifiers are DWIDTH=17.10, AWIDTH=8, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5.

4. The default print of the qualifier table is obtained by

## DBDICT QUALIFIERS

and is equivalent to

```
DBDICT QUALIFIERS ,
  SELECT(KEY QUALALL) ,
  FORMAT(DWIDTH=17.10 AWIDTH=8 IWIDTCH=5 LWIDTCH=1 ,
         FWIDTH=12.5 COLSPACE=2) SORT(KEY=A) ,
  LABEL('QUALIFIERS' CENTER )
```

and looks like:

### Figure 3-3 DBDICT QUALIFIERS Example

QUALALL selects all qualifiers to be printed. The qualifiers will be printed in alphabetic order. QUALSET selects only the qualifiers SEID, PEID, SPC, MPC, LOAD, and METH to be printed. Table 4 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors. QUALALL and QUALSET may not be specified in the FORMAT or SORT descriptors. The qualifier names and values are not printed one per row, but rather from left to right as one logical line that is allowed to wrap after 132 columns.

Table 4 DBDICT QUALIFIERS Colnames

| colname   | Default col-width | Default col-label | Description                          |
|-----------|-------------------|-------------------|--------------------------------------|
| KEY       | 5                 | KEY               | Key number.                          |
| qual-name | See Note.         | qualifier name    | Qualifier name.                      |
| QUALALL   | See Note.         | qualifier name    | All qualifiers.                      |
| QUALSET   | See Note.         | qualifier name    | Predefined subset of all qualifiers. |

**Note:** Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

5. The default print of the current qualifier table is obtained by

DBDICT QUALCURR

and is equivalent to

```
DBDICT QUALCURR SELECT (QUALALL),
FORMAT (AWIDTH=8,IWIDTH=5,LWIDTH=1,COLSPACE=2),
LABEL=('CURRENT QUALIFIERS' CENTER)
```

and looks like:

```
* * * * * D I C T I O N A R Y P R I N T * * * * *
EXECUTION OF DMAP STATEMENT NUMBER 24
MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER 20
                                         CURRENT QUALIFIERS
APRCH B2GG      B2PP      BMETH CMETH CONFIG DEFORM DELTA DESITER DLOAD DRMM DYRD EXTRCV FMETH FREQ FSCOUP
GUST HIGHQUAL HINDEX IC IKBAR IMACHNO IPANEL IQ ISA ISOLAPP K2GG K2PP LOAD M2GG
M2PP MACHINE METH METHF MFLUID MODEL MPC MTEMP NCASE NL99 NLOAD NLOOP NOQUAL OPERALEV OPERASYS
P2G PEID EVALID SDAMP SEDWN SEID SOLAPP SOLID SPC STATSUB SUBDMAP SUBMODEL SUPORT
TEMPLD TFL TSTEP ZNAME ZUZR1 ZUZR2 ZUZR3
'   '   '   '   '   0   0   0   0   0   0   F   0   0   0   0   F   0   0   0   0   0   0   0
'   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0
'   '   0   0   0   0   0   0   0   0   0   0   100   0   0   400   0   0   -1   0   0   0   0
'   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0
```

Figure 3-4 DBDICT QUALCURR Example

Table 5 gives the allowable colnames and a description that may be specified in the SELECT descriptors.

Table 5 DBDICT QUALCURR Colnames

| colname   | Default col-width | Default col-label | Description                          |
|-----------|-------------------|-------------------|--------------------------------------|
| qual-name | See Note.         | qualifier name    | Qualifier name.                      |
| QUALALL   | See Note.         | qualifier name    | All qualifiers.                      |
| QUALSET   | See Note.         | qualifier name    | Predefined subset of all qualifiers. |

**Note:**

Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

6. The default print of the Local Data Block Table is obtained by

DBDICT DATABLK (LOCAL)

and is equivalent to

```
DBDICT DATABLK (LOCAL),
  SELECT (NAME, SUBDMAP, SIZE='BLOCKS', PURGED='PU',
    EQUIVD='EQ', POINTER, TRL1, TRL2, TRL3, TRL4,
    TRL5, TRL6, TRL7),
  FORMAT (NAME=8, SUBDMAP=8, IWIDTH=8, COLSPACE=2),
  SORT (NAME=A) LABEL ('LOCAL DATABLOCKS' CENTER)
```

and looks like:

```
* * * * * D I C T I O N A R Y P R I N T * * * * *
EXECUTION OF DMAP STATEMENT NUMBER      23
MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER      19
                           LOCAL DATABLOCKS
NAME     SUBDMAP     BLOCKS   PU   EQ   POINTER   TRL1   TRL2   TRL3   TRL4   TRL5   TRL6   TRL7
CASEW   PHASE1DR      1       0     0   131780     201      4       0     308      0     0     0
```

Figure 3-5 DBDICT DATABLK(LOCAL) Example

TRL*i* specifies the data block trailer word *i* where  $1 \leq i \leq 10$ . TRAILER selects all 10 data block trailer words.

Table 6 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 6 DBDICT DATABLK(LOCAL) Colnames

| colname | Default col-width | Default col-label | Description               |
|---------|-------------------|-------------------|---------------------------|
| NAME    | 8                 | NAME              | Parameter name.           |
| SUBDMAP | 8                 | SUBDMAP           | SubDMAP name.             |
| SIZE    | 8                 | BLOCKS            | Number of blocks.         |
| EQUIVD  | 8                 | EQ                | Equivalenced flag.        |
| PURGED  | 8                 | PU                | Scratch flag.             |
| POINTER | 8                 | POINTER           | Directory pointer.        |
| TRLi    | 8                 | TRLi              | i-th word in the trailer. |
| TRAILER | 8                 | TRLi              | All 10 trailer words.     |
| EXTNAME | 8                 | EXTNAME           | Extended name.            |

7. The default print of the local parameter table is obtained by

```
DBDICT PARAM(LOCAL)
```

and is equivalent to

```
DBDICT PARAM(LOCAL) SELECT (NAME, SUBDMAP, VALUE),
  FORMAT (COLSPACE=4, VALUE=40, AWIDTH=8),
  SORT (NAME=A) LABEL (' LOCAL PARAMETERS' CENTER)
```

and looks like:

```
* * * * * D I C T I O N A R Y   P R I N T * * * * *
EXECUTION OF DMAP STATEMENT NUMBER      24
  MODULE NAME = DBDICT , SUBMAP SEKRRS , OSCAR RECORD NUMBER    20
                                         LOCAL PARAMETERS
NAME      SUBDMAP      VALUE
-----
AERO      SESTATIC     FALSE
AERO      PHASEIDR    FALSE
ALTRED    SESTATIC     NO
ALTRED    PHASEIDR    NO
ALTSHAPE  SESTATIC     0
ALWAYS    PHASEIDR    -1
ALWAYS    PHASEIC     -1
ALWAYS    SEKRRS      -1
ALWAYS    SESTATIC     -1
APP       PHASEIDR    STATICS
APP       PHASEIC     STATICS
APP       SESTATIC     STATICS
APRCH    SESTATIC
ASING    PHASEIDR    0
ASING    SEKRRS      0
ASING    PHASEIC     0
ASING    SESTATIC     0
```

Figure 3-6 DBDICT PARAM(LOCAL) Example

Table 7 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 7 DBDICT PARAM(LOCAL) Colnames

| colname | Default col-width | Default col-label | Description     |
|---------|-------------------|-------------------|-----------------|
| NAME    | 8                 | NAME              | Parameter name. |
| SUBDMAP | 8                 | SUBDMAP           | SubDMAP name.   |
| VALUE   | 40                | VALUE             | Parameter name. |

8. The default print of Project-Version Table is obtained by

```
DBDICT PROJVERS
```

and is equivalent to

```
DBDICT PROJVERS ,
  SELECT (PROJECT='PROJECT NAME', PROJ='PROJ NO.' ,
    VERS='VERSION', DELFLG='DELETED' ,
    CDATE='CREATION DATE' CTIME='CREATION
    TIME') ,
    FORMAT (PROJECT=40, PROJ=10, VERS=10, DELFLG=7,
      COLSPACE=1 , CDATE=13,CTIME=13) ,
      LABEL ('PROJECT-VERSION',CENTER) ,
      SORT (PROJ=A, VERS=A)
```

and looks like:

```
* * * * * D I C T I O N A R Y P R I N T * * * * *
EXECUTION OF DMAP STATEMENT NUMBER 19
MODULE NAME = DBDICT , SUBMAP SEKRRS , OSCAR RECORD NUMBER 15
                                         PROJECT-VERSION
PROJECT NAME          PROJ NO.   VERSION DELETED CREATION DATE CREATION TIME
-----'-----'-----'-----'-----'-----'
'LEFT FENDER           '        1       1      930805      72319
```

Figure 3-7 DBDICT PROJVERS Example

Table 8 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 8 DBDICT PROJVERS Colnames

| colname | Default col-width | Default col-label | Description                                |
|---------|-------------------|-------------------|--|
| PROJECT | 40                | PROJECT NAME      | Project name defined by PROJECT statement. |
| PROJ    | 10                | PROJ NO           | Project number associated with PROJECT.    |

Table 8 DBDICT PROJVERS Colnames

| colname | Default col-width | Default col-label | Description  |
|---------|-------------------|-------------------|--|
| VERS    | 10                | VERSION           | Version number.  |
| DELFLG  | 7                 | DELETED           | Flag indicating whether this project/version has been deleted by the RESTART NOKEEP or DBCLEAN statements. |
| CDATE   | 13                | CREATION DATE     | Creation date.   |
| CTIME   | 13                | CREATION TIME     | Creation time.   |

CDATE is printed as YYMMDD where YY, MM, and DD are the year, month, and date, respectively. CTIME is HHMMSS where HH, MM, and SS are the hour, minute, and second, respectively.

9. If a parameter or qualifier value is defined to be character string, then the value will be printed with enclosing single quotation marks. Blank strings will also be printed with single quotation marks.
10. If a given qualifier is not in the path of a given data block or parameter, then blank spaces will be printed.
11. A line will wrap if additional columns need to be printed and not enough space is available on the output (assumed to be 132). The first column of each additional line is to be indented by the width of the first column printed for the entry.
12. The where-expr has the following rules:
  - If the where-expr specifies a colname that is not assigned to the data block or parameter, then no directory information will be printed for that data block or parameter. For example, given that SPC is not a qualifier for KGG, the following DBDICT statement will produce no output:
 

```
DBDICT DATABLK=KGG WHERE (SPC=10)
```
  - If the where-expr does not specify a colname that is assigned to the data block (or parameter), then the qualifier is wildcarded. For example, given that SEID is a qualifier for KAA, the following DBDICT statements are equivalent:
 

```
DBDICT DATABLK=KAA
DBDICT DATABLK=KAA WHERE (SEID = *)
```
13. A colname specified in the where-expr must be specified in the SELECT clause if the SELECT clause is also specified.

### Examples:

1. Print the project version table with a title.
 

```
DBDICT PROJVERS SORT (PROJ, VERSION) LABEL ('PROJECT
VERSION TABLE' LEFT)
```
2. Print a directory of all data blocks qualified with PEID=10 or SEID=10. Print columns for the NAME and DBSET, and the qualifiers SPC, MPC, and LOAD.
 

```
DBDICT DATABLK SELECT (NAME, SPC, MPC, LOAD, DBSET, SIZE,
```

```
SEID,PEID) ,  
SORT(NAME,SIZE=D) WHERE( SEID=10 OR PEID=10)
```

## **DBDIR**

Prints Database Directory Tables

Obsolete. See the DBDICT statement.

## DBFIX

### Database Directory Error Detection

Detects and optionally corrects errors in the database directory.

Format: DBFIX [ { LIST } { CORRECT } ]  
          { NOLIST } { NOCORRECT }

#### Example:

DBFIX LIST, NOCORRECT

The preceding example requests a printout of the directory pointers and any errors, but not the corrections.

| Descriptor | Meaning  |
|------------|--|
| LIST       | Requests a debug listing of the database directory pointers. |
| NOLIST     | Suppresses a debug listing of the database directory.        |
| CORRECT    | Corrects the database if any errors are found.               |
| NOCORRECT  | Suppresses the correction of the database.                   |

#### Remarks:

1. It is recommended that a backup copy of the database be made before this statement is used, since corrections of the database are achieved through the deletion of data. Data blocks and parameters are deleted from the database if they have (1) incorrect paths (different than listed in the NDDL); (2) incorrect names (two or more names that are not equivalenced and reference the same data), or (3) incorrect directory pointers.
2. NOLIST does not suppress the listing of any corrections made to the database.

**DBLOAD**

Loads a Database from a FORTRAN File

Recover data blocks or parameters from a database created by the DBUNLOAD statement.

Format:

DBLOAD [DATABLK=  $\left[ \begin{array}{c} * \\ (\text{datablk-list}) \end{array} \right]$ ] PARAM=  $\left[ \begin{array}{c} * \\ (\text{param-list}) \end{array} \right]$  WHERE(where-expr) ,

CONVERT(convert-expr) UNIT = unit FORMAT { BINARY  
NEUTRAL } { OVRWRT  
NOOVRWRT }

Example:

1. Load the database stored in ASCII format on FORTRAN unit 12.

```
DBLOAD      UNIT=12 FORMAT=NEUTRAL
ASSIGN      DBLOAD='physical file name of unloaded database'
            UNIT=12 FORMATTED
```

2. Load version 1 of KAA under project FRONT BUMPER and store it on the primary database under version 5 and project BUMPER. Overwrite duplicates found on the primary database.

```
DBLOAD      DATABLK=(KAA) WHERE (PROJECT='FRONT BUMPER'
            AND, SEID=10 AND VERSION=1) CONVERT(VERSION=5;
            PROJECT='BUMPER') OVRWRT
ASSIGN      DBLOAD='physical file name of unloaded database'
```

| Descriptor   | Meaning  |
|--------------|--|
| datablk-list | Specifies a list of data blocks separated by commas. The default is * which selects all data blocks. The loaded data block may be renamed in the primary database by specifying a slash after the old name, followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.    |
| param-list   | Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The loaded parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified. |

| Descriptor         | Meaning   |
|--------------------|---|
| where-expr         | A logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for more information on WHERE and CONVERT clauses.   |
| convert-expr       | The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is* for all DBsets. The default for qual is *, which is all qualifier values found on the loaded database. See also Remark 8.<br><br>Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is:<br><br>PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name;<br>quali=qual-expri[...]<br><br>For example, CONVERT (SEID=100+SEID; SPC=102). See the beginning of this section for more information on WHERE and CONVERT clauses.<br><br>The default action for VERSION and PROJECT is to use the same version IDs and project IDs; i.e., CONVERT(PROJECT=PROJECT;<br>VERSION=VERSION). But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. The default action for qualifiers and DBSET is to use the same values as long as they are defined in both databases. If not, see Remark 8. |
| unit               | Specifies the FORTRAN unit number of the database to be loaded. The unit must be specified on an ASSIGN statement that references the physical filename of the loaded database. The default is 51.  |
| OVRWRT<br>NOOVRWRT | By default, if duplicate data blocks or parameters exist on the loaded and primary databases, then a fatal message is issued. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as a data block or parameter on the primary database.   |
| NEUTRAL<br>BINARY  | The database to be loaded may be in BINARY or NEUTRAL format. BINARY indicates the database to be loaded is in binary or FORTRAN unformatted format. NEUTRAL indicates the database to be loaded is in ASCII format. The default is BINARY.   |

### Remarks:

1. The DBLOAD statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be loaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be loaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be loaded.

3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. The database to be loaded is attached as read-only. In other words, items can only be fetched and not stored on this database.
5. If more than one DBLOAD statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOAD statements, then the last duplicate will be used.
6. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If the database to be loaded and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
  - If a qualifier in the NDDL of the database to be loaded is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double-precision, complex, or character, then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not loaded.
  - If a DBset-name in the NDDL of the database to be loaded is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the database to be loaded.
9. Data blocks that are equivalenced on the database to be loaded remain equivalenced as long as they are loaded in the same DBLOAD statement or in consecutive DBLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.
10. It is not possible to restart from a database created by DBLOAD in the same run.
11. SOL 190 (or DBTRANS) is also required with DBLOAD if:
  - The database to be loaded has a different BUFFSIZE.
  - The database to be loaded is in neutral format or is being transferred between different machine types.

See also the [Database Concepts in the MSC Nastran Reference Guide](#).

**DBLOCATE**

Attaches Secondary Databases

Obtains data blocks or parameters from prior versions of the primary database, or other databases. DBLOCATE may also be used to compress the primary database and to migrate databases created in prior MSC Nastran versions.

**Format:**

$$\text{DBLOCATE} \left[ \text{DATABLK} = \begin{bmatrix} * \\ (\text{datablk-list}) \end{bmatrix} \right] \text{PARAM} = \begin{bmatrix} * \\ (\text{param-list}) \end{bmatrix} \text{WHERE}(\text{where-expr}) ,$$

$$\text{CONVERT}(\text{convert-expr}) \text{LOGICAL} = \text{dbname} \left[ \begin{bmatrix} \text{OVRWRT} \\ \text{NOOVRWRT} \end{bmatrix} \right] \text{COPY}$$

**Example:**

- Locate in version 4 of MASTER3 all data blocks named KAA for all superelements with IDs greater than 0.

```
DBLOCATE      DATABLK=(KAA) WHERE (PROJECT='FRONT BUMPER' , AND
                                SEID>0 AND VERSION=4) LOGI=MASTER3
ASSIGN        MASTER3='physical file name of master DBset'
```

- Copy all data blocks and parameters from the last version of MASTER3 to the primary database. For all items with the qualifier SEID, change the SEID to twice the old ID number.

```
DBLOCATE      CONVERT (SEID=2*SEID) COPY LOGI=MASTER3
ASSIGN        MASTER3='physical file name of master DBset'
```

- Compress a database with multiple versions. All versions under the current project-ID (see PROJ statement) will be copied from the database OLDDB to NEWDB.

```
ASSIGN        MASTER3=' physical filename of new master DBset'
ASSIGN        OLDDB='physical filename of old master DBset'
DBLOCATE     LOGI=OLDDB COPY WHERE (VERSION=*) ,
              CONVERT (VERSION=VERSION; PROJECT=PROJECT)
```

| Descriptor         | Meaning   |
|--------------------|---|
| datablk-list       | Specifies a list of data blocks separated by commas. The default is *, which selects all data blocks. The located data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.  |
| param-list         | Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The located parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.   |
| where-expr         | A logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example, WHERE(VERSION= 4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for more information on WHERE and CONVERT clauses.<br><br>The default for VERSION is the last version-ID and PROJECT is the current project-ID. The default for qual is *, which is all qualifier values found on the located database. See also Remark 9.  |
| convert-expr       | Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is:<br><br>PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name;<br>quali=qual-expri[;...]<br><br>For example,<br><code>CONVERT (SEID=100+SEID; SPC=102)</code><br>See the beginning of this section for more information on WHERE and CONVERT clauses.<br><br>The default action for VERSION and PROJECT is to convert to the current version-ID and current project-ID. But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. See Example 3. The default action for qualifiers and DBSET is to use the same values as long both databases have the same NDDL schema. If not, see Remark 9. |
| dbname             | Specifies the logical name of the master directory DBset of the located database. dbname must be specified on an ASSIGN statement, which references the physical file name. By default, the located database is also the primary database. (If dbname is specified for the primary database, then dbname must be MASTER.)   |
| OVRWRT<br>NOOVRWRT | By default, duplicate data blocks or parameters on the located database will take precedence over those on the primary database. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as the data block or parameter on the primary database. If NOOVRWRT is specified, then a fatal message is issued.  |
| COPY               | Requests that the located data blocks or parameters be copied to the primary database.  |

**Remarks:**

1. The DBLOCATE statement and its applications are discussed further in the [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be located. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be located. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be located.
3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. If more than one DBLOCATE statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOCATE statements, then the last duplicate will be used.
5. If the located database is not the primary database, then it is attached for read-only purposes. In other words, items can only be fetched and not stored on the located database.
6. If the RESTART FMS statement is also specified, then located data blocks and parameters are treated as if they exist in the restart version. In other words, restart equivalences will be made on located items at the beginning of the run and can be subsequently broken as a result of regeneration and/or NDDL dependencies.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If LOGICAL refers to the primary database and one version is to be copied to another, then the items are equivalenced.
9. If the located database and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
  - If a qualifier in the NDDL of the located database is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double precision, complex or character then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not located.
  - If a dbset-name in the NDDL of the located database is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the located database.

**DBSETDEL**

Deletes a DBset

Deletes a DBset, all of its members, and associated physical files.

**Format:**

DBSETDEL dbsetnamei

**Example:**

Delete DBset DBUP20 from the database.

DBSETDEL DBUP20

| Descriptor | Meaning   |
|------------|---|
| dbsetnamei | Specifies the name(s) of DBset(s) to be deleted. The DBset names MASTER, OBJSCR, or SCRATCH may not be specified. |

**Remarks:**

1. The DBSETDEL statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If dbsetnamei does not exist, then no action is taken.
3. After a DBset has been deleted with this statement, it may be recreated with the INIT statement in a subsequent run.

**DBUNLOAD**

Unloads a Database to a FORTRAN File

Stores data blocks or parameters from the primary database onto a FORTRAN file in a binary or neutral format, for purposes of database compression or database transfer between different computers.

**Format:**

DBUNLOAD [DATABLK= [ \* ] (datablk-list)] PARAM= [ \* ] (param-list) WHERE(where-expr)

UNIT = unit FORMAT = { { BINARY } { NEUTRAL } } { REWIND } { NOREWIND }

**Example:**

1. Unload the database in ASCII format onto FORTRAN unit 12.

|          |   |
|----------|---|
| DBUNLOAD | UNIT=12 FORMAT=NEUTRAL  |
| ASSIGN   | DBUNLOAD='physical file name of FORTRAN unit 12' ,<br>UNIT=12 FORMATTED |

2. Unload version 1 of KAA under project FRONT BUMPER.

|          |   |
|----------|---|
| DBUNLOAD | DATABLK=(KAA) WHERE(PROJECT='FRONT BUMPER' ,AND<br>SEID=10 AND VERSION=1) |
| ASSIGN   | DBUNLOAD=' physical file name of FORTRAN unit 50'                         |

| Descriptor   | Meaning  |
|--------------|--|
| datablk-list | Specifies a list of data blocks separated by commas. The default is*, which selects all data blocks.   |
| param-list   | Specifies a list parameters separated by commas. The default is*, which selects all parameters.  |
| where-expr   | Logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example,<br>WHERE (VERSION=4 AND SEID<>2 AND SEID>0)<br>selects all items under version 4 for all values of SEID greater than 0 except 2.<br>See the beginning of this section on WHERE and CONVERT Clauses. |

| Descriptor         | Meaning  |
|--------------------|--|
|                    | The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is * for all DBsets. The default for qual is *, which is all qualifier values found on the primary database.                                 |
| unit               | Specifies the FORTRAN unit number to unload the database. The unit must be specified on an ASSIGN statement, which references its physical filename. The default is 50.  |
| NEUTRAL<br>BINARY  | The database may be unloaded in BINARY or NEUTRAL format. BINARY indicates the database is to be unloaded in binary or FORTRAN unformatted. NEUTRAL indicates the database is to be unloaded in ASCII format. The default is BINARY. |
| NOREWIND<br>REWIND | By default, if DBUNLOAD is executed more than once for the same unit, then the unit is not rewound. REWIND requests that the unit be rewound prior to unloading.   |

### Remarks:

1. The DBUNLOAD statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be unloaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be unloaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be unloaded.
3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. If more than one DBUNLOAD statement is specified, then they will be processed in the order in which they appear.
5. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
6. If NEUTRAL is specified, then only data blocks with an NDDL description are unloaded. (See the [MSC Nastran DMAP Programmer's Guide](#) under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be unloaded in BINARY if TYPE=UNSTRUCTURED, KDICT, or KELM.
7. Data blocks that are equivalenced on the primary database remain equivalenced as long as they are unloaded in the same DBUNLOAD statement or in consecutive DBUNLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.

**DBUPDATE**

Specifies Database Directory Update Interval

Specifies the maximum length of CPU time between database directory updates to the MASTER DBset. This statement is intended to be used if the INIT MASTER(RAM=r) option is specified.

**Format:**

DBUPDATE [=] update-time

**Example:**

DBUPDATE = 5.5

The preceding example would call for a database directory update at the end of a DMAP module execution after five and one-half minutes of CPU time have elapsed from the last update.

| Descriptor  | Meaning  |
|-------------|--|
| update-time | CPU time interval in minutes (real or integer) between database directory updates. |

**Remarks:**

1. The difference in CPU time from the last update is checked after the execution of each DMAP instruction. The database directory is updated if this difference is greater than update-time. Update-time and CPU time is accurate to the nearest whole second only.
2. If update-time < 0, then database directory updates are only performed at the end of the run.
3. Defaults for update-time are machine dependent and may be found in the *MSC Nastran Installation and Operations Guide*.
4. Periodic updates of the directory tables to MASTER DBset increases the integrity of the database during system crashes (for example, crashes due to insufficient time or space).
5. Directory updates are performed automatically at various points in the execution of the DMAP in addition to those specified by DBUPDATE. An asterisk appears after the word "BEGN" in the executive summary table whenever an update occurs. See the [Output Description](#) in the *MSC Nastran Reference Guide*. These updates occur whenever a permanent data block, parameter DMAP equivalence, or restart equivalence is broken. Updates also occur upon deletions. Additions to the database do not automatically cause a directory update to take place.
6. This statement is in effect only when INIT MASTER(RAM=r) is being used. INIT MASTER(S) and INIT MASTER(NORAM) disable periodic and automatic updates.
7. Update-time may also be changed with the DMAP instruction PUTSYS(update-time, 128) or the NASTRAN SYSTEM(128)=update-time statement. (The update-time must be a real, single-precision value specified in minutes.)

**DEFINE****Parameter Definition**

Assigns user-defined keywords (or cellnames) to a NASTRAN system cell. (See the NASTRAN statement for a description of “cellname”.) In addition, the DEFINE statement provides a mechanism to set default values for system cells.

**Format:**

DEFINE keyword [ =expression ] [ LOCATION=SYSTEM(i) ] [ TYPE=type ]

| Describer  | Meaning   |
|------------|---|
| keyword    | User-defined name, 1 through 24 characters in length. The first character must be alphabetic. The following characters can be used for keywords: A through Z, _, and 0 through 9. Any other characters are invalid.   |
| expression | Expression produces a single value from a set of constant and/or variable parameters separated by operators. The value is assigned to the “keyword” and is also used to set the value for the NASTRAN system cell specified by “LOCATION”. TYPE determines both the type of the result and the type conversions that will be applied to the constants and variables within the expression--mixed mode expressions are allowed (see Remark 6.). The parentheses can be used to change the order of precedence. Operations within parentheses are performed first, with the usual order of precedence being maintained within the parentheses. The variable parameters within the expression must be keywords previously defined on a DEFINE statement. The following operations are allowed: |

| Parameter Type  | Operator | Operation  |
|-----------------|----------|--|
| Integer or Real | +        | Addition   |
|                 | -        | Subtraction  |
|                 | *        | Multiplication   |
|                 | /        | Division   |
| Logical         | +        | Bit-wise OR  |
| Logical         | -        | Bit clear. For example, the result of $a-b$ is equal to the value of $a$ with the bits associated with $b$ set to 0. |

| Descriptor        | Meaning  |             |      |                   |   |      |   |         |         |
|-------------------|--|-------------|------|-------------------|---|------|---|---------|---------|
| SYSTEM(I)<br>type | <p>Specifies the NASTRAN system cell number to be associated with the keyword.</p> <p>The type of expression result, and the type of conversions that will be applied to the constants and variables within the expression. Allowable data types are as follows:</p> |             |      |                   |   |      |   |         |         |
|                   | <table border="1"> <thead> <tr> <th>Description</th><th>Type</th></tr> </thead> <tbody> <tr> <td>Integer (default)</td><td>I</td></tr> <tr> <td>Real</td><td>R</td></tr> <tr> <td>Logical</td><td>LOGICAL</td></tr> </tbody> </table>                                | Description | Type | Integer (default) | I | Real | R | Logical | LOGICAL |
| Description       | Type   |             |      |                   |   |      |   |         |         |
| Integer (default) | I  |             |      |                   |   |      |   |         |         |
| Real              | R  |             |      |                   |   |      |   |         |         |
| Logical           | LOGICAL  |             |      |                   |   |      |   |         |         |

**Remarks:**

1. If TYPE, LOCATION, and EXPRESSION are omitted, the default data type is Integer and the default value is zero.
2. If EXPRESSION is omitted, an internal default will be assigned to the keyword/cellname based on the LOCATION (See [The NASTRAN Statement](#) for a list of internal default values).
3. A DEFINE statement that specifies a LOCATION is actually setting the default for a NASTRAN system cell, and therefore it is not necessary to also set the system cell value on a subsequent NASTRAN statement unless the user wishes to override the previous DEFINE statement setting. Also, since more than one DEFINE statement may be present for the same “keyword”, the last specification takes precedence. “Keywords” referenced on a NASTRAN statement, or in an expression on the DEFINE statement, are automatically substituted by the last specification of the “keyword” prior to the current statement being processed.
4. DEFINE statements may also be specified in runtime configuration (RC) files. See the [MSC Nastran 2020 Installation and Operations Guide](#).
5. System cells may also be set with the NASTRAN statement. In addition, they may be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See the [MSC Nastran DMAP Programmer’s Guide](#).
6. Each operand within the expression will be converted to the result type prior to the arithmetic operation. For example: the statement “DEFINE JJ=2.5 + 3.6 TYPE=I” would result in 2.5 and 3.6 being converted to 2 and 3, respectively, and the result of 5 would be assigned to JJ.

**Examples:**

1. Change the default value for block size:  
`DEFINE BUFFSIZE=4097 LOCATION=SYSTEM(1)`
2. Set the sparse matrix selection to forward-backward substitution only:  
`DEFINE SPARSE=16 LOCATION=SYSTEM(126)`

3. Define the system cell keyword and default value for the maximum output line count and then reset it to another value on a NASTRAN statement. Note: The DEFINE statement would typically be placed in an RC file and the NASTRAN statement would be placed in the File Management Section whenever the user wants to override the DEFINE statement default setting.

```
DEFINE      MAXLINES=999999999 LOCATION=SYSTEM (9)
NASTRAN    MAXLINES=100000
```

4. Define system cells that behave like “toggles,” turning some feature on or off:

```
DEFINE      MESH=2 LOCATION=(31)
DEFINE      NOMESH=0 LOCATION=(31)
NASTRAN    MESH
```

**Note:**

Since each subsequent DEFINE statement redefines the default value, the second DEFINE of system cell location 31 sets the default value to 0. A NASTRAN statement can then be inserted in the input file to reset the MESH system cell back to a value of 2. This same technique can be used with any system cell where the user wishes to refer to the system cell keyword and have the system cell set to a previous DEFINE statement default.

5. Invalid usage of the DEFINE and NASTRAN statement:

```
DEFINE      BUFSIZE=4097
NASTRAN    BUFSIZE=2048
```

Valid usage:

```
DEFINE      BUFSIZE=4097 LOCATION=SYSTEM(1)
NASTRAN    BUFSIZE=2048
```

**ENDJOB**      Terminates Job

Terminates the job at a user-specified location in the FMS Section.

**Format:**

ENDJOB

**Example:**

DBDICT  
ENDJOB

**Remark:**

ENDJOB is normally used after a DBDICT statement, or after database initialization.

**EXPAND****Concatenates New DBset Members**

Concatenates additional DBset members on an existing permanent DBset previously defined with an INIT statement.

**Format:**

EXPAND dbset-name LOGICAL=( log-namei [(max-sizei)]...)

**Example:**

|        |                              |
|--------|------------------------------|
| ASSIGN | DBMEM02='physical file name' |
| EXPAND | DBALL LOGICAL=(DBMEM02)      |

This would create and add the DBset member DBMEM02 to the existing DBset DBALL.

| Descriptor | Meaning   |
|------------|---|
| dbset-name | The name of a DBset previously defined with an INIT statement.  |
| log-namei  | Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement which refers to the physical file name of the DBset member. |

| Descriptor   | Meaning   |
|--------------|---|
| max-size     | Specifies the maximum size in blocks, words or bytes of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the following unit keywords: |
| Unit Keyword | Storage Unit  |
| W            | Words   |
| B            | Bytes   |
| KW, K        | Kilowords (1024 words)  |
| KB           | Kilobytes (1024 bytes)  |
| MW, M        | Megawords ( $1024^2$ words)   |
| MB           | Megabytes ( $1024^2$ bytes)   |
| GW, G        | Gigawords ( $1024^3$ words)   |
| GB           | Gigabytes ( $1024^3$ bytes)   |
| TW, T        | Terawords ( $1024^4$ words)   |
| TB           | Terabytes ( $1024^4$ bytes)   |

For example, 100MB = 100 megabytes, 1.5GB = 1.2 gigabytes = 1536 megabytes, and 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFFSIZE.

#### Remark:

1. On all computers with dynamic file allocation, the physical filename of a DBset member may be specified on an ASSIGN statement:

```
ASSIGN log-name='physical filename'
```

If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in [Database Concepts in the MSC Nastran Reference Guide](#).

**INCLUDE**

Inserts External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

INCLUDE 'filename'

**Example:**

The following INCLUDE statement is used to obtain the bulk data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

| Descriptor | Meaning  |
|------------|--|
| filename   | Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (''). |

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file:

/dir123/dir456/dir789/filename.dat

may be included with the following input:

```
INCLUDE '/dir123
          /dir456
          /dir789/filename.dat'
```

3. See the [MSC Nastran 2020 Installation and Operations Guide](#) for more examples.

**INIT**                      Creates a DBset

Creates a temporary or permanent DBset. For the SCRATCH and MASTER DBsets, all or some of their space may be allocated to real memory.

**Format 1: Initialize any DBset except MASTER and SCRATCH:**

INIT    DBset-name    [LOGICAL=(log-namei(max-sizei),...)    BUFFSIZE=b    CLUSTER=c]

**Format 2: Initialize the MASTER DBset:**

INIT MASTER  $\left[ \begin{array}{l} \text{RAM = r}, s \\ \text{NORAM} \end{array} \right]$  LOGICAL = (log-name(max-sizei), ...),  
BUFFSIZE = b   CLUSTER = c ]

**Format 3: Initialize the SCRATCH DBset:**

INIT SCRATCH  $\left[ \begin{array}{l} \text{MEM = m} \\ \text{NOMEM} \end{array} \right]$  LOGICAL = (log-name(max-sizei), ...),  
SCR300 = (log-namei(max-sizei),...)   BUFFSIZE = b   CLUSTER=c ]

**Example:**

1. Modify the default allocation of the DBALL DBset to 50000 blocks:

INIT DBALL LOGI=(DBALL(50000))

2. Do not allocate any real memory for the MASTER and SCRATCH DBsets:

INIT                      MASTER (NORAM)  
INIT                      SCRATCH (NOMEM)

3. Create a new DBset called DBUP with two members DBUP1 and DBUP2:

INIT                      DBUP LOGI=(DBUP1, DBUP2)  
ASSIGN                    DBUP1 ='physical filename 1'  
ASSIGN                    DBUP2='physical filename 2'

| Descriptor | Meaning  |
|------------|--|
| dbset-name | The name of a temporary or permanent DBset.  |
| MASTER     |  |
| SCRATCH    |  |
| log-namei  | Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement, which refers to the physical file name of the DBset member. If no log-namei is specified, then the DBset will have one member and the log-name will be the same as the DBset-name. A maximum of twenty log-names may be specified. For the SCRATCH DBset, see also Remark 8. SCR300 is a special keyword that indicates that the log-names are members reserved for DMAP module internal scratch files. |
| max-sizei  | Specifies the maximum size, in blocks, words, or bytes, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the following unit keywords:   |

| Unit Keyword | Storage Unit                |
|--------------|-----------------------------|
| W            | Words                       |
| B            | Bytes                       |
| KW, K        | Kilowords (1024 words)      |
| KB           | Kilobytes (1024 bytes)      |
| MW, M        | Megawords ( $1024^2$ words) |
| MB           | Megabytes ( $1024^2$ bytes) |
| GW, G        | Gigawords ( $1024^3$ words) |
| GB           | Gigabytes ( $1024^3$ bytes) |
| TW, T        | Terawords ( $1024^4$ words) |
| TB           | Terabytes ( $1024^4$ bytes) |

For example, 100MB = 100 megabytes, 1.5GB = 1.5 gigabytes = 1536 megabytes, and 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFSIZE. The default for DBALL and SCRATCH may be found in the [MSC Nastran 2020 Installation and Operations Guide](#) and ranges from 250,000 blocks to 4,000,000 blocks.

RAM=r requests that r words of real memory are to be allocated for the MASTER DBset. See the [nast20200 Command](#) (p. 14) in the *MSC Nastran Reference Guide*. The default is RAM or RAM=30000. NORAM or RAM=0 specifies that no real memory is to be allocated.

| Descriptor | Meaning  |
|------------|--|
| S          | If the primary database is being created in the run, this option requests that all DBsets in the primary database will be automatically deleted at the end of the run. INIT MASTER(S) is equivalent to specifying scr=yes on the “nastran” statement. See <a href="#">Executing MSC Nastran, 1</a> . If the run is a restart, then this option is ignored.                                       |
| MEM        | MEM=m specifies that m blocks of real memory are to be allocated for the SCRATCH DBset. See <a href="#">The NASTRAN Statement (Optional)</a> (p. 14) in the <i>MSC Nastran Reference Guide</i> . The default m is machine dependent and may be found in the <a href="#">MSC Nastran 2020 Installation and Operations Guide</a> . NOMEM or MEM=0 requests that no real memory is to be allocated. |
| NOMEM      |  |
| BUFFSIZE   | BUFFSIZE=b specifies the number of words per block in the DBset and will override the value specified by the BUFFSIZE keyword on the NASTRAN statement. The default for b is obtained from the NASTRAN BUFFSIZE statement. See the <i>MSC Nastran 2020 Installation and Operations Guide</i> .   |
| CLUSTER    | CLUSTER=c specifies the number of blocks per cluster in the DBset. The default is 1, and any other value is not recommended.   |

**Remarks:**

1. The INIT statement and its applications are further discussed in [Database Concepts in the MSC Nastran Reference Guide](#).
2. Four DBsets are predefined and automatically allocated by the program. Their DBset-names are MASTER, DBALL, SCRATCH, and OBJSCR, and they are described in [Database Concepts in the MSC Nastran Reference Guide](#).
3. On all computers with dynamic file allocation, the physical filename of a DBset member may be specified on an ASSIGN statement:  
  

```
ASSIGN log-name='physical filename'
```

If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in [Database Concepts in the MSC Nastran Reference Guide](#).
4. It is recommended that there be sufficient physical space to hold a DBset member should it reach its maximum size (max-sizei). The max-sizei may be converted to words by multiplying by b. A summary of space usage and allocation is printed at the end of the execution summary table.
5. In restart runs, the INIT statement is ignored for preexisting permanent DBsets. The INIT statement is intended to be specified only in the run in which the DBset is being created. If more DBset members need to be added to the DBset, then the EXPAND statement is used.
6. If RAM or RAM=r is specified and the run terminates because the computer operating system detects insufficient space or time, or the computer halts due to a power outage or operator interruption, then it may not be possible to restart from the database. See the DBUPDATE FMS statement.

7. BUFFSIZE=b and CLUSTER=c must satisfy the following inequality:

$$b \leq \frac{64000}{c} + 5$$

8. By default, the SCRATCH DBset is divided into two partitions: LOGICAL and SCR300. The LOGICAL partition, log-names after the LOGICAL keyword, are reserved for DMAP scratch data blocks, and the SCR300 partition for DMAP module internal scratch files.
  - The maximum total number of log-names for LOGICAL and SCR300 is 20. For example, if LOGICAL has 8 log-names, then SCR300 can have no more than 12 log-names.
  - If NASTRAN SYSTEM(142)=1 is specified, then the SCR300 partition is not created and internal scratch files, along with DMAP scratch data blocks, will reside on the LOGICAL partition. The default is SYSTEM(142)=2.
  - If NASTRAN SYSTEM(151)=1 is specified and the LOGICAL partition has reached its maximum size, then the SCR300 partition will be used. The default is SYSTEM(151)=0.
  - By default, the space specified for the SCR300 partition is released to the operating system after each DMAP module is executed if the module used more than 100 blocks for internal scratch files. If 100 blocks is not a desirable threshold, then it may be changed by specifying NASTRAN SYSTEM(150)=t, where t is the number of blocks for the threshold.
9. BUFFSIZE=b is predefined for DBset-names MSCOBJ, OBJSCR, and USROBJ and may not be changed by BUFFSIZE on this statement or if the NASTRAN BUFFSIZE=bmax statement (see [The NASTRAN Statement \(Optional\)](#) (p. 14) in the *MSC Nastran Reference Guide*). The default for b is recommended for all except very large problems. bmax must reflect the maximum of b specified for all DBsets attached to the run, including the delivery database. See [MSC Nastran 2020 Installation and Operations Guide](#) for the defaults of b and bmax.
10. If INIT MASTER(RAM=r) and INIT SCRATCH(MEM=m) are specified, then BUFFSIZE for these DBsets must be the same. If not, a warning message is issued, and the BUFFSIZE for the SCRATCH DBset is reset to that of the MASTER DBset.
11. Only one INIT statement per dbset-name may be specified in the File Management Section.

**MEMLIST****Specify Datablocks Eligible for SMEM**

Specifies a list of scratch datablocks that may reside in scratch memory (SMEM).

**Format:**

MEMLIST DATABLK = (DBname1, DBname2, ..., DBnamei)

**Example:**

MEMLIST DATABLK = (KOO, MOO, KQQ, MQQ)

If generated, datablocks KOO, MOO, KQQ, and MQQ will reside in scratch memory. All other datablocks will be excluded from scratch memory.

| Descriptor | Meaning                      |
|------------|------------------------------|
| DBnamei    | Name of a Nastran datablock. |

**Remarks:**

1. Only NDDL and local scratch datablocks may be included in MEMLIST specification.
2. Datablocks specified will reside in SMEM on a first-come, first-served basis.
3. Datablocks not specified by this statement will not reside in SMEM.
4. Database directories for the SCRATCH DBset reside in SMEM and are not affected by any MEMLIST specification.
5. Continuation lines are allowed.
6. Multiple MEMLIST statements are honored.
7. Scratch I/O activity is reported in the .f04 file by including DIAG 42 in the Executive Control Section.

**PROJ**      Defines Database Project-Identifier

Defines the current or default project identifier, project-ID.

**Format:**

PROJ [=] 'project-ID'

**Examples:**

1. PROJ = 'MY JOB'
2. The following project-ID will be truncated to 40 characters:

PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B RUN'

and all subsequent restarts must have the statement.

PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B'

| Descriptor | Meaning  |
|------------|--|
| project-ID | Project identifier. Must be enclosed in single quotes. (Character string, maximum of 40 characters; Default=blank) |

**Remarks:**

1. There may be only one PROJECT statement in the File Management Section. The PROJECT statement must be specified before all DBCLEAN, DBDICTIONARY, RESTART, DBLOCATE, and DBLOAD statements where project-ID is not specified by the user.
2. This statement is optional and specifies that all data blocks and parameters to be stored on or accessed from the database in the current run shall also be identified by project-ID. Therefore, in subsequent runs that may access this data through other FMS statements such as RESTART, the project-ID must be specified.
3. Project-ID is the default on DBCLEAN, DBDICTIONARY, and RESTART FMS statements and in the WHERE and CONVERT clause of the DBLOCATE statement.
4. Leading blanks and trailing blanks enclosed within the single quotes are ignored. All other blanks are considered part of the project-ID.
5. Project-ID is saved with only the first 40 characters specified.

**RESTART**

## Reuses Database From a Previous Run

Requests that data stored in a previous run be used in the current run.

**Format:**

```
RESTART [ PROJECT='project' VERSION = [version-ID] [KEEP  
LAST] [NOKEEP] ] LOGICAL = dbname ]
```

**Examples:**

1. RESTART VERSION=7

Version number 7 will be retrieved for this run (version 8). At the end of the run, version 7 will be deleted.

2. PROJ='FENDER'

RESTART

The last version under project-ID FENDER will be used in the current run.

3. ASSIGN RUN1='run1.MASTER'

RESTART LOGICAL=RUN1

The run1.MASTER and its associated database will be used (read only) for restart purposes.

| Descriptor | Meaning   |
|------------|---|
| project-ID | Project identifier. See description of the PROJ FMS statement. Must be enclosed in single right-hand quotation marks (') (Character string, maximum of 40 characters; default is the project-ID specified on the PROJ FMS statement).   |
| version-ID | Version number (Integer > 0).   |
| LAST       | Specifies the last version under project-ID.  |
| KEEP       | Data stored under VERSION will remain on the database after the run is completed.   |
| NOKEEP     | Data stored under VERSION will be deleted from the database after the run is completed.   |
| dbname     | Specifies the logical name of an existing MASTER (master directory) DBset to be used for restart purposes. This MASTER and its associated database will be opened in a read-only mode to perform the restart; any new data will be written to the database for the current run. |

**Remarks:**

1. There may only be one RESTART statement in the File Management Section.
2. A new version-ID is automatically assigned whenever a restart is performed.

3. If project-ID or version-ID or both are specified and cannot be found, then a fatal message will be issued.
4. The RESTART statement is required to perform restarts in solution sequences 4, and 101 through 200 and SOL 400.
5. If PROJECT is not specified, then the run will restart from the project-ID specified on the PROJ statement (See Example 2.).
6. Databases created in one version typically cannot be directly restarted into a different version. Restrictions are typically documented in the current release guide; however, a DBLOCATE type restart might work.
7. Restarts do not work with DMP. Restarts do not work with ACMS for versions released before 2018.
8. If NASTRAN IFPSTAR=YES (default) and scr=no then the Bulk Data images will be stored on the IFPDAT file; for example, if the input file name is run1 then run1.IFPDAT will be created along with run1.MASTER and run1.DBALL. Please note the following when performing a restart.
  - a. If run1.IFPDAT, run1.MASTER and run1.DBALL are in the same folder then only the MASTER file needs to be specified on an ASSIGN statement. For example, ASSI cold='run1.MASTER' RESTART LOGI=cold
  - b. If run1.IFPDAT is in a different folder than that of run1.MASTER and run1.DBALL (or IFPDAT has been renamed in the same folder) then both the MASTER and IFPDAT must be specified on ASSIGN statements and the IMPORT keyword must be specified on the ASSIGN IFPDB statement:  
ASSI cold='run1.MASTER'  
RESTART LOGI=cold  
ASSI IFPDB='other\_folder/run1.IFPDAT' IMPORT
9. For SOL 400 one must also use [NLRESTART \(Case\)](#).
10. The database is not suitable for restart if any of the following was specified in the cold start. If a restart is attempted then the program will issue User Fatal Message 9061.
  - a. EXTSEOUT(DMIGDB or MATDB or MATRIXDB) without defining DBEXT.
  - b. scr=post
  - c. scr=no and "NASTRAN SYSTEM(316)>0".



# 4

## Executive Control Statements

- Key to Descriptions 108
- Executive Control Statement Descriptions 110

## Key to Descriptions

Braces { } indicate that a choice of describers is mandatory.

A brief sentence about the function of the statement is given.

Brackets [ ] indicate that a choice of describers is optional.

### LINK

Link a Main SubDMAP

The default describers are shaded.

**Format:**

LINK [ <sup>n</sup> subDMAP-name ] SOLO = SOLOUT-DBset EXEOUT = exeout-DBset

Describers in uppercase letters are keywords that must be specified as shown.

INCLUDE = incl-DBset [ MAP NOMAP ], SOLNAME = newname

Describers in lower case are variables.

#### Describers:

**n** The solution number of the main description for the list of valid numbers. (Integer > 0).

**subDMAP-name** The name of a main subDMAP. See the *MD Nastran DMAP Programmer's Guide*.

**solout-DBset** The name of a DBset in the solution sequence map. If the describer is in lower case, then it is a variable and the describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

**exeout-DBset** The name of an alternate DBset. Only the OSCAR map is supported.

**incl-DBset** The name of a DBset included in the solution sequence. See Remark 2.

**newname** A new name assigned to the OSCAR. The default is newname.

Each of the describers is discussed briefly.

**MAP** Further details may be discussed under Remarks. Give the name of all the subsequence.

**NOMAP** Suppresses printing of the link map.

#### Remarks:

- All DBsets specified on this statement must have the same BUFSIZE. See the INIT FMS statement.
- SubDMAP objects are created with the current run or obtained from previous runs. The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the describers.
  - Objects created with the COMPILE statement in the current run.
  - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.

## Executive Control Section

This section describes the Executive Control statements. These statements select a solution sequence and various diagnostics.

Most Executive Control statements are order independent. The exceptions are the COMPILE, COMPILER, ALTER, ENDALTER, and LINK statements. If used, the LINK statement must appear after all COMPILE statements. The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. The COMPILER statement also sets the defaults for subsequent COMPILE statements.

## Executive Control Statement Summary

The Executive Control statements are summarized as follows:

|                              |  |
|------------------------------|--|
| <a href="#">ALTER</a>        | Specifies deletion and/or insertion of the DMAP statements that follow.            |
| <a href="#">APP</a>          | Specifies an approach in a solution sequence.                                      |
| <a href="#">CEND</a>         | Designates the end of the Executive Control statements.                            |
| <a href="#">COMPILE</a>      | Requests compilation of specified subDMAPs or the NDDL file.                       |
| <a href="#">COMPILER</a>     | Specifies DMAP compilation diagnostics.  |
| <a href="#">DIAG</a>         | Requests diagnostic output or modifies operational parameters.                     |
| <a href="#">DOMAINSOLVER</a> | Selects domain decomposition solution methods.                                     |
| <a href="#">ECHO</a>         | Controls the echo of Executive Control statements.                                 |
| <a href="#">ENDALTER</a>     | Designates the end of a DMAP sequence headed by an ALTER.                          |
| <a href="#">GEOMCHECK</a>    | Specifies tolerance values and options for optional finite element geometry tests. |
| <a href="#">ID</a>           | Specifies a comment.   |
| <a href="#">LINK</a>         | Requests the link of a main subDMAP.   |
| <a href="#">MALTER</a>       | Inserts and/or deletes DMAP statements in solution sequences                       |
| <a href="#">MODEL_CHECK</a>  | Specifies model checkout run options.  |
| <a href="#">SOL</a>          | Requests execution of a solution sequence or DMAP program.                         |
| <a href="#">SOL 600, ID</a>  | Creates Marc input and optionally executes Marc from SOL 600                       |
| <a href="#">SOL 700, ID</a>  | Executes MSC Nastran Explicit Nonlinear (SOL 700)                                  |
| <a href="#">SPARSEsolver</a> | Specifies various options used in sparse solution if equations operations.         |
| <a href="#">TIME</a>         | Sets the maximum allowable execution time.   |

## Executive Control Statement Descriptions

Executive Control statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

### Description

A brief sentence about the function of the statement is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
COMPILE   SEDRCVR   SOUIN=MSCSOU,  
        NOREF      NOLIST
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), its allowable range, and its default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks in the remarks section are generally arranged in order of importance and indicate such things as the Executive Control statement's relationship to other statements, restrictions and recommendations on its use, and further descriptions of the describers.

**\$ Comment**

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

**Format:**

\$ followed by any characters out to column 80.

**Example:**

\$ TEST FIXTURE-THIRD MODE

**Remarks:**

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the bulk data.

**ALTER****Inserts and/or Deletes DMAP Statements**

Inserts and/or deletes DMAP statements in a subDMAP.

**Format:**

```
ALTER k1 [,k2]
```

or

```
ALTER 'string1' [(occurrence,offset)] ,['string2' [(occurrence,offset)] ]
```

or

```
ALTER k1 , ['string2' [(occurrence,offset)] ]
```

or

```
ALTER 'string1' [(occurrence,offset)] , [k2]
```

**Examples:**

1. The following alter will insert a MATPRN DMAP statement after the first occurrence of the string 'SDR2' in subDMAP DSASTAT:

```
SOL 101
COMPILE DSASTAT $
ALTER 'SDR2' $
MATPRN OESDS1//$
CEND
```

2. The following alter will delete the second occurrence of the OUTPUT4 DMAP statement in subDMAP DSASTAT and replace it with a MATPRN DMAP statement:

```
SOL 101
COMPILE DSASTAT $
ALTER 'OUTPUT4' (2) , 'OUTPUT4' (2) $
$ OR
$ ALTER 'OUTPUT4' (2) , '' $
MATPRN OESDS1//$
CEND
```

| Descriptor | Meaning  |
|------------|--|
| k1         | If k2 or 'string2' is not specified, the subsequent DMAP statements will be inserted after either the statement number k1 or the 'string1', [(occurrence,offset)] reference point. |
| k1, k2     | DMAP statements numbered k1 through k2 will be deleted and may be replaced with subsequent DMAP statements.  |
| 'string1'  | if 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.  |

| Descriptor          | Meaning  |
|---------------------|--|
| 'string1','string2' | DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequent DMAP statements.             |
| occurrence          | This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP (Integer > 0; Default = 1).   |
| offset              | This flag indicates the offset from the reference DMAP statement.<br>Depending on the sign the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement (Integer; Default = 0). |

### Remarks:

1. The ALTER statement must be used in conjunction with the COMPILE Executive Control statement. Note: ALTER statements cannot be used in conjunction with an MALTER statement, and therefore should never immediately follow this statement.
2. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.
3. The ALTERs can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
  - K2 or 'string2'(occurrence, offset) references must refer to a DMAP statement number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single ALTER statement.
  - K1 or 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another ALTER that references the same subDMAP.
4. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement; i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string—all blanks and comments either embedded or immediately preceding the DMAP statement, will be retained. However, comments are ignored for the following type of alter:  
`alter '^ *gp0'`
5. Within a SUBDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP—not at the current position of the last string match.
6. The special characters (metacharacters) used for string searching are described in Remark 9. The characters <, >, and \$, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string  
`IF (DDRMM >=-1)`  
 the command is  
`ALTER 'IF (DDRMM \>=-1)' $`
7. The ALTER statement must not exceed 72 characters (no continuations are allowed).

8. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (""). For example, the alter statement

```
ALTER 'string1' (r1,01), ''
```

is equivalent to

```
ALTER 'string1' (r1,01), 'string1' (r1,01)
```

The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

As another example, the alter statement

```
ALTER 'string1' (r1,01), '' (r2,02)
```

is equivalent to

```
ALTER 'string1' (r1,01), 'string1' (r2,02)
```

9. Metacharacters\*:

|              |   |
|--------------|---|
| .            | Matches any single character except newline.  |
| *            | Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since “.” (dot) means any character, “.*” means “match any number of characters”.  |
| [...] or < > | Matches any one of the characters enclosed between the brackets. For example, “[AB]” matches either “A” or “B”. A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example “[A-Z]” will match any uppercase letter from A to Z and “[0-9]” will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character not in the list. |
| ^ or ! or .  | Requires that the following regular expression be found at the beginning of the line. Note that these metacharacters may lead to User Fatal Message 802 if the preceding line is a comment.   |
| \$           | Requires that the preceding regular expression be found at the end of the line.   |
| \            | Treats the following special character as an ordinary character. For example, “\.” stands for a period and “\*” for an asterisk. Also, to search for a tic (’), the search string must be single quotes.  |
| ,            | Marks the beginning and end of a pattern to be matched.   |
| Note:        | Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ ->! and [ ] -><>) if portability is required. However, all the preceding characters are recognized by MSC Nastran.   |

10. If a string-based alter uses the “!” in the expression (find occurrence at the beginning of line), it is possible MSC Nastran will fail with User Fatal Message 802.

**APP****Specifies Solution Sequence Approach**

Selects heat transfer analysis in the linear static solution sequence SOL 101, or a coupled analysis combining heat transfer and structural analysis in SOL 153.

**Format:**

APP approach

**Example:**

The following requests a heat transfer rather than a structural analysis in SOL 101.

SOL 101  
APP HEAT

| Descriptor | Meaning  |
|------------|--|
| approach   | Specifies one of the following:  |
| HEAT       | Indicates that heat transfer is to be performed in SOL 101.  |
| COUPLED    | Indicates that a coupled analysis combining heat transfer and structural analysis is to be preformed in SOL 153. |

**Remarks:**

1. The APP statement is optional.
2. The APP HEAT statement applies only to linear static SOL 101. The APP HEAT statement is not required in SOLs 153 and 159, or in SOL 101 if PARAM,HEATSTAT,YES is specified.
3. The NASTRAN HEAT=1 statement is an alternate specification of APP HEAT. See [Executing MSC Nastran, 1](#).

**CEND** End of Executive Control Section Delimiter

Designates the end of the Executive Control Section.

**Format:**

CEND

**Remark:**

1. CEND is an optional statement. If CEND is not specified, then the program will automatically insert one.

**COMPILE**

Compiles DMAP Statements

Requests the compilation of a subDMAP, subDMAP alter, or NDDL sequence.

**Format 1:** Compiles a subDMAP or subDMAP alter sequence

COMPILE [SUBDMAP  
DMAP] subDMAP-name [SOUIN = souin-DBset SOUOUT = souout-DBset,

OBJOUT = objout-DBset [LIST] [REF] [DECK]  
[NOLIST] [NOREF] [NODECK]

**Format 2:** Compiles an NDDL sequence

COMPILE NDDL = nddl-name { { SOUIN = souin-dbset } { LIST } } { { SOUOUT = souout-dbset } { NOLIST } } { { REF } { NOREF } } { { DECK } { NODECK } }

**Examples:**

1. The following compiles an alter in subDMAP PHASEIDR:

```
COMPILE PHASE1DR
ALTER 'CALL PHASE1A'
CEND
```

2. The following compiles a subDMAP called MYDMAP. (SUBDMAP and END are DMAP statements; see the [MSC Nastran DMAP Programmer's Guide](#)):

```
COMPILE MYDMAP LIST REF
SUBDMAP MYDMAP $
.
.
.
END $
CEND
```

3. The following obtains a listing of the NDDL:

```
ACQUIRE NDDL
COMPILE NDDL=NDDL LIST
CEND
```

| Descriptor   | Meaning  |
|--------------|--|
| subDMAP-name | The name of a subDMAP sequence. SubDMAP-name must be 1 to 8 alphanumeric characters in length and the first character must be alphabetic. The keywords DMAP and SUBDMAP are optional and do not have to be specified.  |
| nndl-name    | The name of an NDDL sequence (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The keyword NDDL must be specified.   |
| souin-DBset  | The name of a DBset from which the subDMAP or NDDL source statements will be retrieved (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is MSCSOU if the next statement is not a subDMAP statement. |
| souout-DBset | The name of a DBset on which the subDMAP or NDDL source statements will be stored (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is the SCRATCH DBset.  |
| objout-DBset | The name of a DBset on which the subDMAP object code will be stored (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is the OBJSCR DBset.   |
| LIST,NOLIST  | LIST requests a compiled listing of the subDMAP or NDDL sequence. NOLIST suppresses the listing. NOLIST is the default.  |
| REF,NOREF    | REF requests a compiled cross reference of the subDMAP or NDDL sequence. NOREF suppresses the cross reference. NOREF is the default.   |
| DECK,NODECK  | DECK requests the subDMAP or NDDL source statements to be written to the PUNCH file. NODECK suppresses the writing to the PUNCH file. NODECK is the default.   |

### Remarks:

1. SubDMAP names for MSC Nastran solution sequences are given in the SOL statement description. The “COMPLIER LIST REF” statement may be used to determine the appropriate subDMAP-name.
2. If a subDMAP is being compiled and SOUIN=souin-DBset is specified, then an ALTER Executive Control statement, or an INCLUDE statement which contains an ALTER statement as the first non-comment line, must appear immediately after this statement. If not, then the SUBDMAP DMAP statement must appear immediately after this statement. See the [MSC Nastran DMAP Programmer’s Guide](#).
3. Starting in MSC/Nastran Version 69, DBsets USRSOU and USROBJ were no longer automatically created. They must be initialized by the INIT FMS statement and then may be specified for souin-dbset (or souout-dbset) and objout-dbset, respectively. They may be used to store the subDMAP source statements and object code on the primary database for reexecution in a subsequent run. Consider the following example:

In the first run, the following COMPILE statement compiles and stores a subDMAP called MYDMAP:



```
COMPILE MYDMAP SOUOUT=USRSSOU OBJOUT=USROBJ
SUBDMAP MYDMAP $
.
.
END $
CEND
```

In the second run, the SOL statement is used to execute the MYDMAP stored in the previous run. The LINK statement is required to retrieve the object code from the USROBJ DBset:

```
SOL MYDMAP
LINK MYDMAP INCL=USROBJ
CEND
```

In the third run, the COMPILE statement is used to alter MYDMAP and execute:

```
SOL MYDMAP
COMPILE MYDMAP SOUIN=USRSSOU
ALTER...
.
.
.
CEND
```

4. If SOUOUT or OBJOUT is specified and a subDMAP with the same name as subDMAP-name already exists on the database, then its source statements or object code will be replaced.
5. A COMPILE statement is required for each subDMAP to be compiled. If two or more COMPILE statements reference the same subDMAP name, then only the last is used in the linking of the object code. If the COMPILE statement is being used only to alter a subDMAP and two or more COMPILE statements reference the same subDMAP name, then the multiple alters are assembled and the subDMAP is compiled only once.
6. Only one COMPILE statement for an NDDL sequence may be specified in the input file.
  - SOUIN=souin-DBset requests only a compilation of the NDDL sequence stored on souin-DBset for purposes of obtaining a listing or a cross reference, and it cannot be modified with the ALTER statement. See Remark 3. COMPILE NDDL=NDDL SOUIN=MSCSOU LIST requests a listing of the MSC Nastran NDDL sequence. The ACQUIRE FMS statement or the SOL statement must be specified in order to attach the corresponding delivery database.
  - To alter the MSC Nastran NDDL sequences, the entire modified NDDL sequence is included after the COMPILE statement, and SOUIN=souin-DBset is not specified.
  - SOUOUT=souout-DBset requests the storage of the NDDL source statements on the souout-DBset, and may not be specified with SOUIN=souin-DBset.
7. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, and NODECK. In other words, if LIST or NOLIST, REF or NOREF, or DECK or NODECK is not specified, then the corresponding option on the COMPILER statement will be used. In the following example, REF on the COMPILER statement will override the default of NOREF on the COMPILE statement:

```
COMPILER REF
```

```
COMPILE MYDMAP
```

8. MSCSOU and MSCOBJ, specified with SOUOUT and OBJOUT, are special DBsets similar to USRSOU and USROBJ except that they are used in the creation or modification of a delivery database. For an example application, see the [MSC Nastran 2020 Installation and Operations Guide](#).

**COMPILER****DMAP Compiler Output Options**

Requests compilation of a DMAP sequence and/or overrides defaults on the COMPILE statement.

**Format:**

```
COMPILER[=][ LIST ][ DECK ][ REF ][ GO ][ SORT ]
      [ NOLIST ] [ NODECK ] [ NOREF ] [ NOGO ] [ NOSORT ]
```

**Example:**

COMPILER=LIST

| Describer       | Meaning   |
|-----------------|---|
| LIST, NOLIST    | LIST requests the compilation listing of the solution sequence. NOLIST suppresses the listing.  |
| DECK,<br>NODECK | DECK requests that the DMAP source statements of the solution sequence be written to the PUNCH file. NODECK suppresses the DECK option. |
| REF, NOREF      | REF requests a compilation cross reference. NOREF suppresses a compilation cross reference.   |
| GO, NOGO        | GO requests the execution of the solution sequence following compilation. NOGO requests termination following compilation.              |
| SORT,<br>NOSORT | SORT compiles subDMAPs in alphabetical order. NOSORT compiles subDMAPs in calling sequence order.                                       |

**Remarks:**

1. REF is equivalent to DIAG 4. LIST is equivalent to DIAG 14. DECK is equivalent to DIAG 17.
2. NOGO is an alternative to NOEXE on the SOL statement.
3. This statement provides a means of obtaining a compilation or source listing, or both, of a complete solution sequence, including all the component subDMAPs.
4. See the COMPILE statement to compile a single subDMAP.
5. This statement also requests the automatic link of the solution sequence. Therefore, all objects must be created in the current run or obtained from the DBset such as USROBJ. See the COMPILE statement for how to create and store objects.
6. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, NODECK on the COMPILE entry when they are not explicitly specified. However, COMPILER LIST produces a list of the entire solution sequence. If a listing of only specific subdmaps is desired, then COMPILER LIST should not be specified and the LIST request should be made on the COMPILE entry.

```
COMPILER REF
COMPILE MYDMAP
```

**DIAG****Requests Diagnostic Output**

Requests diagnostic output or special options.

**Format:**

DIAG [=] k1[k2, ..., kn]

**Examples:**

DIAG 8, 53

or

DIAG 8

DIAG 53

| Descriptor | Meaning  |
|------------|--|
| ki         | A list separated by commas and/or spaces of desired diagnostics. |

**Remarks:**

1. The DIAG statement is optional.
2. Multiple DIAG statements are allowed.
3. The following table lists the possible values for ki and their corresponding actions:

|     |   |
|-----|---|
| k=1 | Dumps memory when a nonpreface fatal message is generated.  |
| k=2 | Prints database directory information before and after each DMAP statement. Prints bufferpooling information.   |
| k=3 | Prints “DATABASE USAGE STATISTICS” after execution of each functional module. This message is the same as the output that appears after the run terminates. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .   |
| k=4 | Prints cross-reference tables for compiled sequences. Equivalent to the COMPILER REF statement.   |
| k=5 | Prints the BEGIN time on the operator’s console for each functional module. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .   |
| k=6 | Prints the END time for each functional module in the log file or day file, and on the operator’s console. Modules that consume less time than the threshold set by SYSTEM(20) do not create a message. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> . |
| k=7 | Prints eigenvalue extraction diagnostics for the complex determinate method.  |
| k=8 | Prints matrix trailers as the matrices are generated in the execution summary table. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .  |

- k=9 Prints a message in the .f04 file when EQUIV and EQUIVX perform a successful equivalence; in other words, both the input and output exists.
- k=10 Selects an alternate option for averaging nonlinear loading (NOLINI Bulk Data entry) in linear transient analysis. Replaces  $N_{n+1}$  with  $(N_{n+1} + N_n + N_{n-1})/3$ .
- k=11 DBLOAD, DBUNLOAD, and DBLOCATE diagnostics.
- k=12 Prints eigenvalue extraction diagnostics for complex inverse power and complex Lanczos methods.
- k=13 Prints the open core length (the value of REAL). See the [Output Description in the MSC Nastran Reference Guide](#).
- k=14 Prints solution sequence. Equivalent to the COMPILER LIST statement.
- k=15 Prints table trailers.
- k=16 Traces real inverse power eigenvalue extraction operations
- k=17 Punches solution sequences. Equivalent to the COMPILER DECK statement.
- k=18 In aeroelastic analysis, prints internal grid points specified on SET2 Bulk Data entries.
- k=19 Prints data for MPYAD and FBS method selection in the execution summary table.
- k=20 Similar to DIAG 2 except the output appears in the execution summary table and has a briefer and more user-friendly format. However, the .f04 file will be quite large if DIAG 20 is specified with an MSC Nastran solution sequence. A DMAP alter with DIAGON(20) and DIAGOFF(20) is recommended. DIAG 20 also prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics. See the [MSC Nastran DMAP Programmer's Guide](#).
- k=21 Prints diagnostics of DBDICTIONARY and DBENTRY table.
- k=22 EQUIV and EQUIVX module diagnostics.
- k=23 Not used.
- k=24 Prints files that are left open at the end of a module execution. Also prints DBVIEW diagnostics.
- k=25 Outputs internal plot diagnostics.
- k=27 Prints Input File Processor (IFP) table. See the [MSC Nastran Programmer's Manual](#), Section 4.5.9.
- k=28 Punches the link specification table (XBSD). The Bulk Data and Case Control Sections are ignored, and no analysis is performed.
- k=29 Process link specification table update. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.
- k=30 In link 1, punches the XSEMii data (i.e., sets ii via DIAG 1 through 15). The Bulk Data and Case Control Sections are ignored, and no analysis is performed. After link 1, this turns on BUG output. Used also by MATPRN module. See also Remark 5 on the [TSTEP, 3285](#) Bulk Data entry.

- k=31 Prints link specification table and module properties list (MPL) data. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.
- k=32 Prints diagnostics for XSTORE and PVA expansion.
- k=33 Not used.
- k=34 Turns off plot line optimization.
- k=35 Prints diagnostics for 2-D slideline contact analysis in SOLs 106 and 129.
- k=36 Prints extensive tables generated by the GP0 module in p-version analysis.
- k=37 Disables the superelement congruence test option and ignores User Fatal Messages 4277 and 4278. A better alternative is available with PARAM,CONFAC. See [Parameters, 783](#).
- k=38 Prints material angles for CQUAD4, CQUAD8, CTRIA3, and CTRIA6 elements. The angle is printed only for elements that use the MCID option of the connection entry.
- k=39 Traces module FA1 operations and aerodynamic splining in SOLs 145 and 146.
- k=40 Print constraint override/average information for edges and faces in p-adaptive analysis
- k=41 Traces GINO OPEN/CLOSE operations.
- k=42 Prints output on .f04 file the usage statistic for datablock defined in the FMS command, MEMLIST.
- k=43 Print output for Buffer Pool Diagnostics.
- k=44 Prints a mini-dump for fatal errors and suppresses user message exit.
- k=45 Prints the same database directory information as DIAG 2 except that it prints only after each DMAP statement.
- k=46 Used by MSC Nastran development for GINO printout.
- k=47 Prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics.
- k=48 Used by MSC Nastran development for GINO printout.
- k=49 DIAG 49 is obsolete and should not be used. The utility f04rprt should be used to summarize the .f04 execution summary instead.
- k=50 Traces the nonlinear solution in SOLs 106, 129, 153, and 159. Prints subcase status; echoes NLPARM, NLPCI, and entry fields; and prints initial arc-length. Prints iteration summary only in SOLs 129, and 159.  
In static aeroelastic analysis (SOL 144), prints transformation information associated with the generation of the DJX matrix in the ADG module and intermediate solutions information in the ASG module.
- k=51 Prints intermediate displacement, load error vectors, and additional iteration information helpful in debugging in SOLs 106, 129, 153, and 159.
- k=52 Disables the printing of errors at each time step in SOLs 129 and 159.

- k=53      MESSAGE module output will also be printed in the execution summary table. See the [Output Description in the MSC Nastran Reference Guide](#).
- k=54      Linker debug print.
- k=55      Performance timing.
- k=56      Extended print of execution summary table (prints all DMAP statements and RESTART deletions). See the [Output Description in the MSC Nastran Reference Guide](#).
- k=57      Executive table (XDIRLD) performance timing and last-time-used (LTU) diagnostics.
- k=58      Data block deletion debug and timing constants echo.
- k=59      Buffpool debug printout.
- k=60      Prints diagnostics for data block cleanup at the end of each module execution in subroutines DBCLN, DBEADD, and DBERPL.
- k=61      GINO block allocator diagnostics.
- k=62      GINO block manager diagnostics.
- k=63      Prints each item checked by the RESTART module and its NDDL description.
- k=64      Requests upward compatibility DMAP conversion from Version 65 only. Ignored in Version 70.5 and later systems.

**DOMAINsolver**

## Domain Decomposition Solution Method

Selects domain decomposition solution methods.

Format:

$$\begin{aligned}
 & \text{DOMAINsolver} \left[ \begin{array}{l} \text{STAT} \\ \text{MODES} \\ \text{FREQ} \\ \text{ACMS} \\ \text{DSA} \\ \text{NLSOLV} \end{array} \right] \left( \text{PARTOPT} = \left[ \begin{array}{l} \text{DOF} \\ \text{GRID} \\ \text{FREQ} \\ \text{ELEM} \end{array} \right], \text{NUMDOM} = \text{int}, \right. \\
 & \quad \left. \text{UPFACT} = \text{real}, \text{PRINT} = \left[ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], \text{NCLUST} = \text{int}, \text{CLUSTSZ} = \text{int}, \text{COMPMETH} = \left[ \begin{array}{l} \text{SUPER} \\ \text{GRID} \\ \text{NONE} \end{array} \right], \right. \\
 & \quad \left. \text{RUNOPT} = \left[ \begin{array}{l} \text{MSTSLV} \\ \text{MULTIMST} \end{array} \right], \text{PARTMETH} = \left[ \begin{array}{l} \text{BEND} \\ \text{MSCMLV} \\ \text{METISG} \\ \text{METISO} \end{array} \right], \text{TIPSIZE} = \text{int}, \text{VERSION} = \left[ \begin{array}{l} \text{NEW} \\ \text{OLD} \end{array} \right], \text{SCHED} = \left[ \begin{array}{l} \text{STATIC} \\ \text{DYNAMIC} \end{array} \right], \text{GRPSIZ} = \left[ \begin{array}{l} 0 \\ N \\ -1 \end{array} \right] \right)
 \end{aligned}$$

Examples:

```

DOMAINsolver STAT (PARTOPT=DOF)
DOMAINsolver ACMS (UPFACT=3.0, NUMDOM=128)
DOMAINsolver DSA
DOMAINsolver STAT, NLSOLV
DOMAINsolver ACMS (PARTMETH=BEND, NUMDOM=128)
DOMAINsolver ACMS (PARTMETH=METISO, TIPSIZE=500)
DOMAINsolver STAT (PARTMETH=MSCMLV)
DOMAINsolver ACMS (VERSION=NEW) - default
DOMAINsolver ACMS (VERSION=OLD)

```

The keywords function as follows:

|        |  |
|--------|--|
| STAT   | Linear statics.                              |
| MODES  | Normal modes.                                |
| FREQ   | Frequency response.                          |
| ACMS   | Automated component modal synthesis.         |
| DSA    | Design sensitivity analysis. (See Remark 2.) |
| NLSOLV | Nonlinear analysis                           |

The descriptions of the parameters are as follows:

|                   |   |
|-------------------|---|
| PARTOPT           | Partitioning option. Selects which domain is to be decomposed.  |
| DOF               | Degree of freedom domain.   |
| GRID              | Grid point (geometric) domain.  |
| FREQ              | Frequency domain.   |
| ELEM              | Finite element domain.  |
|                   | The default is dependent upon solution sequence. See <a href="#">Table 4-1</a> for further descriptions.  |
|                   | Note that ACMS (VERSION=NEW) requires PARTOPT=DOF. If it is not set by the user, ACMS will use it automatically. However, if the user specifies a different value for PARTOPT, for example, ACMS (VERSION=NEW, PARTOPT=GRID), then MSC Nastran will execute old ACMS rather than new ACMS.  |
| NUMDOM<br>(ACMS)  | Selects the number of domains as follows:<br>If NUMDOM = 0 or 1, then the model will not be split. Default depends on the model size and the value of PARTOPT.<br><br>NUMDOM will be ignored for ACMS (VERSION=NEW); instead of the number of domains, new ACMS uses the size of the tip domains being specified via TIPSIZE (see below). |
| NUMDOM<br>(STAT)  | Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (equal to the number of processors used for the run). The model will be divided into NUMDOM domains in either the geometric (grid-based) or DOF domains, depending on the value of PARTOPT.  |
| NUMDOM<br>(MODES) | Default = <i>dmp</i> ; if NUMDOM has any other value, it will be reset to <i>dmp</i> (equal to the number of processors used for the run). The model will be divided into NUMDOM domains in either the frequency, geometric (grid-based) or DOF domains, depending on the value of PARTOPT.   |
| NUMDOM<br>(FREQ)  | Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (equal to the number of processors used for the run). The frequency range will be divided into NUMDOM regions which are then solved independently.   |
| UPFACT<br>(ACMS)  | By default, the frequency range used for upstream component modes is two times larger than the desired range on the EIGR/L entry. To modify this factor, specify the UPFACT parameter (Real; Default=2.0).  |
| PRINT<br>(ACMS)   | Controls intermediate print of upstream and data recovery processing in .f06 and .f04 files. Default='NO'. If PRINT=NO and an error occurs upstream, the intermediate output is placed in a separate output file named "jid.acms_out" for examination.  |

|                    |  |
|--------------------|--|
| NCLUST<br>(MODES)  | Specifies the number of frequency segments for hierachic parallel Lanczos. The frequency range is divided into NLCUST segments and, if PARTOPT=DOF, the stiffness and mass matrices are partitioned into dmp/NCLUST matrix domains. If PARTOPT = GRID, or if PARTOPT is not specified, the model geometry is partitioned into dmp/NCLUST domains.  |
| CLUSTSZ<br>(MODES) | Specifies the number of matrix or geometric domains for hierachic parallel Lanczos. If PARTOPT=DOF, the stiffness and mass matrices are partitioned into CLUSTSZ matrix domains. If PARTOPT = GRID, or if PARTOPT is not specified, the model geometry is partitioned into CLUSTSZ domains. In either case, the frequency range is divided into dmp/CLUSTSZ frequency segments.  |
| COMP METH          | Specifies the compression method used before domain decomposition.<br>COMP METH is considered only for keywords STAT, MODES and ACMS with PARTOPT = DOF, for all other settings of the keyword and parameter PARTOPT, COMP METH will be ignored. The following are valid options for COMP METH:<br><br>SUPER      Specifies the SUPER compression method. (Default)<br>GRID        Specifies the GRID compression method.<br>NONE        Specifies that no compression should be done.   |
| RUNOPT             | Specifies the DMP run option. The following are valid options for RUNOPT:<br><br>MSTSLV     Specifies "master-slave" DMP execution mode. Only the Master DMP process executes the full Nastran solution. This is optimal for single-host DMP execution.<br><br>MULTIMST    Specifies "multiple-master" DMP execution mode. All DMP processes execute the full Nastran solution. This is optimal for multiple-host DMP execution.<br><br>Nastran detects single- and multiple-host DMP execution and sets the default run option accordingly. |
| PARTMETH           | Partitioning option (characher string). PARTMETH selects which method is used for domain decomposition.<br><br>BEND        Specifies the Extreme/BEND partitioning method<br>MSCMLV     Specifies the MSCMLV partitioning method<br>METISG      Specifies the METIS partitioning method based on number of tip domains requested (see NUMDOM above)  |

|         |   |
|---------|---|
| METISO  | Specifies the METIS partitioning method based on the desired tip size (see TIPSIZE below). Available for ACMS only.   |
|         | <b>Note:</b> NUMDOM (see above) will be ignored for PARTMETH=METISO.  |
|         | <b>Note:</b> ACMS (VERSION=NEW) requires PARTMETH=METISO. If this is not set by the user, ACMS will use it automatically. If the user sets a different value for PARTMETH when ACMS (VERSION=NEW) is also specified, then the MSC Nastran will automatically re-set PARTMETH to METISO and run with new ACMS. |
| TIPSIZE | Specifies a desired number of nodes in the tip domains (integer). ACMS only. This is used only when PARTMETH=METISO; otherwise, it is ignored.<br><br>Default: 1000 (ACMS Version=OLD);<br>200 (ACMS Version=NEW)   |
| VERSION | Selects new ACMS (MSC Nastran Version 2017 +) or old ACMS (prior to MSC Nastran Version 2017). The default value of VERSION is 'NEW'.   |
| SCHED   | For ACMS parallel scheduling with VERSION=NEW. Default is DYNAMIC, which is generally more efficient. Select STATIC for static parallel scheduling. See Remark 7.   |
| GRPSIZ  | Specifies parallel group size for ACMS Dynamic scheduling. Integer:<br><br>0      Default<br>N      Set parallel group size to "N" SMP threads.<br>-1     Use single parallel group; number of threads in the group is NSMP (Number of SMP).  |

Table 4-1 Analysis Types vs Supported Partitioning Methods

| Solution Sequence | DMP Method | Partitioning Methods Available |     |      |          |      |
|-------------------|------------|--------------------------------|-----|------|----------|------|
|                   |            | GRID                           | DOF | FREQ | DOF+FREQ | ELEM |
| 101               | STAT       | *                              | *   |      |          |      |
| 103               | MODES      | *                              | *   | *    |          |      |
|                   | ACMS(OLD)  | *                              | *   |      |          |      |
|                   | ACMS(NEW)  |                                | *   |      |          |      |
| 108               | FREQ       |                                |     | *    |          |      |
| 110               | MODES      |                                |     | *    |          |      |
|                   | ACMS       |                                | *   |      |          |      |
| 111               | MODES      |                                | *   | *    |          |      |
|                   | ACMS(OLD)  | *                              | *   | *    | *        |      |
|                   | ACMS(NEW)  |                                | *   |      |          |      |
|                   | FREQ       |                                |     | *    |          |      |
| 112               | MODES      |                                | *   |      |          |      |
|                   | ACMS       |                                | *   |      |          |      |
| 200               | MODES      |                                | *   | *    | *        |      |
|                   | ACMS       |                                | *   | *    | *        |      |
|                   | FREQ       |                                |     | *    |          |      |
|                   | DSA        |                                |     |      |          |      |
| 400               | STAT       |                                | *   |      |          |      |
|                   | NLSOLV     |                                |     |      |          | *    |
|                   | ACMS       |                                | *   |      |          |      |

\* - means "supported".

The DOMAINSOLVER command is optional. If "dmp=" is specified on the command without a DOMAINSOLVER command in the Executive Control Section, the actions shown in [Table 4-2](#) will result based on solution sequence.

Table 4-2 DOMAINSOLVER Defaults with “dmp=” on command line

| Solution Sequence | Default DOMAINSOLVER Options |                     |
|-------------------|------------------------------|---------------------|
|                   | DMP Method                   | Partitioning Option |
| 101               | STAT                         | DOF                 |
| 103               | MODES                        | DOF                 |
| 108               | FREQ                         | FREQ                |
| 110               | MODES                        | FREQ                |
| 111               | MODES                        | FREQ                |
|                   | FREQ                         | FREQ                |
| 112               | MODES                        | FREQ                |
| 200               | MODES                        | DOF                 |
|                   | FREQ                         | FREQ                |
| 400               | STAT                         | DOF                 |
|                   | NLSOLV                       | ELEM                |

Note that DOMAINSOLVER ACMS is not the default for the relevant solution sequences, but it is the recommended option for larger models for SOL 103, 111, 112, and 200 to obtain the best performance.

#### Remarks:

1. Grid Point Weight Generator output selected by PARAM,GRDPNT or the WEIGHTCHECK Case Control command is not available when PARTOPT=GRID.
2. In SOL 200, design sensitivity calculations may be performed in a distributed parallel environment in SOL 200 with the DSA keyword. It is a coarse parallel implementation that divides the sensitivity task across a number of processors so that each processes a subset of the total number of design variables. Following the sensitivity analysis and before optimization, the separate sensitivity data are appended into a global sensitivity set. Also,
  - a. The dmp=n keyword must be specified on the Nastran submittal command where n is the number of available processors.
  - b. The ACMS keyword may also be specified or modified along with DSA. For example,  
DOMAINSOLVER DSA, ACMS
3. In SOL 400, the STAT method is used for DMP parallelism of matrix factorization and the NLSOLV method is used for DMP parallelism of the NLEMG process. The NLSOLV method is intended for nonlinear analysis using advanced nonlinear methods. To only use the STAT method or to change its option to use, for example, a different compression method, specify the following DOMAINSOLVER command:

DOMAINSOLVER STAT(COMP METH=GRID)

Note that if STAT is changed as specified above, then the DMP parallelism for NLEMG will be turned off. If you want to change the STAT as described above and also keep NLSOLV for NLEMG parallelism, then use the following DOMAINsolver command:

```
DOMAINsolver NLSOLV,STAT(COMPBMETH=GRID)
```

Note that this can equivalently be written as follows:

```
DOMAINsolver NLSOLV
```

```
DOMAINsolver STAT(COMPBMETH=GRID)
```

Lastly, to only use the NLSOLV method for NLEMG parallelism, specify the following option:

```
DOMAINsolver NLSOLV
```

4. In SOL 400, the DOMAINsolver NLSOLV (RUNOPT=MULTIMST) is not supported in combination with Intel MKL Pardiso Solver option, i.e., SPARSEsolver NLSOLV(FACTMETH=PRDLDL).
5. In SOL 111, the MODES or ACMS option is used for DMP parallelism of the modal calculations and the FREQ option is used for the DMP parallelism of the frequency response calculations. The DOMAINsolver FREQ is always turned on for SOL 111. To use the ACMS option for the modal calculation, use the following DOMAINsolver command:

```
DOMAINsolver ACMS
```

Furthermore, if the MODES option needs to be changed for SOL 111, then the correct approach is to include the following DOMAINsolver option:

```
DOMAINsolver MODES(PARTOPT=GRID).
```

6. Residual vectors are not available when PARTOPT=GRID.
7. Dynamic parallel scheduling is available for ACMS and VERSION=NEW. Dynamic scheduling provides better parallel speedup for larger numbers of threads. For NSMP less than or equal to 2, static scheduling is used regardless of the setting of SCHED.

## ECHO

### Controls Printed Echo

Controls the echo (printout) of the Executive Control Section.

#### Formats:

ECHOOFF  
ECHOON

#### Remarks:

1. The ECHO statement is optional.
2. ECHOOFF suppresses the echo of subsequent Executive Control statements. ECHOON reactivates the echo after an ECHOOFF statement.

**ENDALTER**      End of DMAP Alter

Designates the end of an alter.

**Format:**

ENDALTER

**Remark:**

1. The ENDALTER statement is required when using an alter unless the alter package ends with a CEND, COMPILE, or LINK statement.

**GEOMCHECK**

Specifies Geometry Check Options

Specifies tolerance values and options for optional finite element geometry tests.

Format:

GEOMCHECK test\_keyword [= tol\_value], [MSGLIMIT = n],  $\left[ \begin{array}{l} \text{MSGTYPE} = \text{INFORM} \\ \text{FATAL} \\ \text{WARN} \end{array} \right]$

[SUMMARY], [ADVNLELM], [NONE]

Examples:

- Set the tolerance for the CQUAD4 element skew angle test to 15.0 degrees and limit messages to 50:

GEOMCHECK Q4\_SKEW=15.0, MSGLIMIT=50

- Limit messages to 500 for each element type:

GEOMCHECK MSGLIMIT=500

- Set the message type to fatal for CQUAD4 element taper tests:

GEOMCHECK Q4\_TAPER, MSGTYPE=FATAL

- Request summary table output only using default tolerance values:

GEOMCHECK SUMMARY

- Request advanced and conventional element geometry check output using default tolerance values:

GEOMCHECK ADVNLELM

| Descriptor   | Meaning  |
|--------------|--|
| test_keyword | A keyword associated with the particular element geometry test. See Remark 2. for a list of acceptable selections.                     |
| tol_value    | Tolerance value to be used for the specified test. See Remark 2. for default values of the test tolerances.                            |
| n            | The minimum number of messages that will be produced. The default is 100 messages for each element type. See Remark 3.                 |
| FATAL        | Geometry tests that exceed tolerance values produce fatal messages. See Remark 4.  |
| INFORM       | Geometry tests that exceed tolerance values produce informative messages. See Remark 4.  |
| WARN         | Geometry tests that exceed tolerance values produce warning messages. See Remark 4.  |
| SUMMARY      | A summary table of the geometry tests performed is produced. No individual element information messages are output.                    |
| ADVNLELM     | Geometry check performed on Advanced and conventional elements. If no ADVNLELM, geometry check only performed on conventional element. |
| NONE         | None of the optional element geometry tests will be performed.   |

**Remarks:**

1. The GEOMCHECK statement controls the number and severity of certain informational and warning messages produced by element matrix generation geometry checking operations. Controls are currently available for the CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, CTETRA, CPYRAM, CBAR, and CBEAM elements only. Multiple GEOMCHECK statement may be present. Continuations are acceptable.
2. The following table summarizes the acceptable specifications for test\_keyword.

| Name     | Value Type      | Default | Comment                                     |
|----------|-----------------|---------|---|
| Q4_SKEW  | Real $\geq 0.0$ | 30.0    | Skew angle in degrees.                      |
| Q4_TAPER | Real $\geq 0.0$ | 0.50    | Taper ratio.                                |
| Q4_WARP  | Real $\geq 0.0$ | 0.05    | Surface warping factor.                     |
| Q4_IAMIN | Real $\geq 0.0$ | 30.0    | Minimum interior angle in degrees.          |
| Q4_IAMAX | Real $\geq 0.0$ | 150.0   | Maximum interior angle in degrees.          |
| T3_SKEW  | Real $\geq 0.0$ | 10.0    | Skew angle in degrees.                      |
| T3_IAMAX | Real $\geq 0.0$ | 160.0   | Maximum interior angle in degrees.          |
| TET_AR   | Real $\geq 0.0$ | 100.0   | Longest edge to shortest edge aspect ratio. |
| TET_EPLR | Real $\geq 0.0$ | 0.50    | Edge point length ratio.                    |
| TET_EPIA | Real $\geq 0.0$ | 150.0   | Edge point included angle in degrees.       |
| TET_DETJ | Real            | 0.0     | $ J $ minimum value.                        |
| TET_DETG | Real            | 0.0     | $ J $ minimum value at vertex point.        |
| HEX_AR   | Real $\geq 0.0$ | 100.0   | Longest edge to shortest edge aspect ratio. |
| HEX_EPLR | Real $\geq 0.0$ | 0.50    | Edge point length ratio.                    |
| HEX_EPIA | Real $\geq 0.0$ | 150.0   | Edge Point Included Angle in degrees.       |
| HEX_DETJ | Real            | 0.0     | $ J $ minimum value.                        |
| HEX_WARP | Real $\geq 0.0$ | 0.707   | Face warp coefficient.                      |
| PEN_AR   | Real $\geq 0.0$ | 100.0   | Longest edge to shortest edge aspect ratio. |
| PEN_EPLR | Real $\geq 0.0$ | 0.50    | Edge point length ratio.                    |
| PEN_EPIA | Real $\geq 0.0$ | 150.0   | Edge point included angle in degrees.       |
| PEN_DETJ | Real            | 0.0     | $ J $ minimum value.                        |
| PEN_WARP | Real $\geq 0.0$ | 0.707   | Quadrilateral face warp coefficient.        |
| PYR_AR   | Real $\geq 0.0$ | 100.0   | Longest edge to shortest edge aspect ratio. |
| PYR_EPLR | Real $\geq 0.0$ | 0.50    | Edge point length ratio.                    |
| PYR_EPIA | Real $\geq 0.0$ | 150.0   | Edge point included angle in degrees.       |

| Name     | Value Type      | Default | Comment                              |
|----------|-----------------|---------|--------------------------------------|
| PYR_DETJ | Real            | 0.0     | $ J $ minimum value.                 |
| PYR_WARP | Real $\geq 0.0$ | 0.707   | Quadrilateral face warp coefficient. |
| BEAM_OFF | Real $\geq 0.0$ | 0.15    | CBEAM element offset length ratio.   |
| BAR_OFF  | Real $\geq 0.0$ | 0.15    | CBAR element offset length ratio.    |

where:

- Test\_keyword names starting with the characters Q4 are applicable to CQUAD4 and CQUADR elements. Test\_keyword names starting with the characters T3 are applicable to CTRIA3 and CTRIAR elements. Test\_keyword names starting with the characters TET\_ are applicable to CTETRA elements. Test\_keyword names starting with the characters HEX\_ are applicable to CHEXA elements. Test\_keyword names starting with the characters PEN\_ are applicable to CPENTA elements. Test\_keyword names starting with the characters PYR\_ are applicable to CPYRAM elements.
- Skew angle for the quadrilateral element is defined to be the angle between the lines that join midpoints of the opposite sides of the quadrilateral. Skew angle for the triangular element is defined to be the smallest angle at any of the three vertices.
- Interior angles are defined to be the angles formed by the edges that meet at the corner node of an element. There are four for quadrilateral shapes and three for triangular shapes.
- Taper ratio for the quadrilateral element is defined to be the absolute value of [the ratio of the area of the triangle formed at each corner grid point to one half the area of the quadrilateral minus 1.0]. The largest of the four ratios is compared against the tolerance value. Note that as the ratio approaches 0.0, the shape approaches a rectangle.
- Surface warping factor for a quadrilateral is defined to be the distance of the corner points of the element to the mean plane of the grid points divided by the average of the element diagonal lengths. For flat elements (such that all of the grid points lie in a plane), this factor is zero.
- The edge point length ratio and edge point included angle tests are only performed for the solid elements when edge node points exist. The length ratio test evaluates the relative position of the edge node point along a straight line connecting the two vertex nodes of that edge. Ideally, the edge point should be located on this line at a point midway between the two end points. The default tolerance allows the edge node to be positioned anywhere between the two quarter points on this line. In addition, the angle between the lines joining the edge node and the end points is determined. If the angle is less than the tolerance (default is 150°), then the interior angle test is considered violated and a diagnostic message will be generated if appropriate.
- The face warp coefficient test tolerance is the cosine of the angle formed between the normal vectors located at diagonally opposite corner points on each face surface. This value is 1.0 for a face where all four corners lie in a plane. The default tolerance allows angles of up to 45° before a message is generated.

3. A single line of output summarizing the results of all tests for an element will be output if any of the geometry tests exceeds the test tolerance. Only the first n of these messages will be produced. A summary of the test results indicating the number of tolerances exceeded, as well as the element producing the worst violation, is also output. If the SUMMARY keyword has been specified, only the summary table is produced and none of the single line element messages will be output.
4. When SUMMARY is not specified, each geometry test that exceeds the tolerance will be identified in the single line output summary by an indicator based on the specification for MSGTYPE. For the FATAL option, the indicator is “FAIL”; for the INFORM option, it is “xxxx”; for the WARN option, it is “WARN”. If the FATAL option is specified and any test fails, the run is terminated.

**ID**      Comment

Specifies a comment.

Format:

ID [=] i1, i2

| Describer | Meaning   |
|-----------|---|
| i1, i2    | Character strings (1 to 8 characters in length and the first character must be alphabetic). |

Remark:

1. The ID statement is optional and not used by the program.

**INCLUDE****Inserts External File**

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

```
INCLUDE 'filename'
```

**Example:**

The following INCLUDE statement is used to obtain the bulk data from another file called MYEXEC.DATA:

```
SOL 101
INCLUDE 'MYEXEC.DATA'
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
ENDDATA
```

| Descriptor | Meaning   |
|------------|---|
| filename   | Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks ('). |

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file

/dir123/dir456/dir789/filename.dat

may be included with the following input:

```
INCLUDE '/dir123
          /dir456
          /dir789/filename.dat'
```

3. See the [MSC Nastran 2020 Installation and Operations Guide](#) for more examples.

**LINK****Links a Main SubDMAP**

Links a main subDMAP to form a solution sequence.

**Format:**

LINK  $\left\{ \begin{array}{c} n \\ \text{subDMAP-name} \end{array} \right\}$  [SOLOUT = solout-DBset EXECOUT - exeout-DBset,

INCLUDE - incl-DBset  $\left[ \begin{array}{c} \text{MAP} \\ \text{NOMAP} \end{array} \right]$  SOLNAME = newname ]

**Examples:**

1. LINK STATICS

Links the STATICS main subDMAP. The program links any subDMAPs compiled in this run, with any other subDMAP objects called in STATICS and stored on the MSCOBJ DBset.

2. LINK MYDMAP, SOLNAM=STATICS, SOLOUT=USROBJ, NOMAP, INCLUDE=USROBJ

Links MYDMAP and renames the solution sequence executable to STATICS. The executable will be saved on the USROBJ DBset. The order of search for subDMAP objects is:

- Compiled subDMAP in this run.
- USROBJ DBset.

| Descriptor   | Meaning   |
|--------------|---|
| n            | The solution number of the main subDMAP. See the SOL statement description for the list of valid numbers (Integer > 0).   |
| subDMAP-name | The name of a main subDMAP. See the <a href="#">MSC Nastran DMAP Programmer's Guide</a> (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).   |
| solout-DBset | The name of a DBset where the solution sequence executable and the link table of the solution sequence may be stored. See Remark 6. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). |
| exeout-DBset | The name of an alternate DBset different than solout-DBset where only the solution sequence executable may be stored. See Remark 6. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). |
| incl-DBset   | The name of a DBset where other subDMAP objects are obtained. See Remark 2. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).   |

| Descriptor | Meaning   |
|------------|---|
| newname    | A new name which is referenced by the SOL statement. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic; default is subDMAP-name.) |
| MAP        | Prints the link map. A link map will give the name of all the subDMAPs that make up the solution sequence.  |
| NOMAP      | Suppresses printing of the link map.  |

**Remarks:**

1. All DBsets specified on this statement must have the same BUFFSIZE. See the [INIT, 98](#) FMS statement.
2. SubDMAP objects are created with the COMPILE statement either in the current run or obtained from previous runs. The LINK statement collects objects in the following order:
  - Objects created with the COMPILE statement in the current run.
  - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.
3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.
4. The LINK statement must appear after all the COMPILE packages, if any. A compile package begins with the COMPILE statement and is delimited by the ENDALTER, CEND, LINK, or another COMPILE statement.
5. The link table is necessary for COMPILER (or DIAG 4, 14, 17) Executive Control statement requests and the automatic link process.
6. EXEOUT is useful in building delivery databases where executables are not to be saved. EXEOUT will be defaulted to the same DBset as specified by SOLOUT.

**MALTER****Inserts and/or Deletes DMAP Statements in Solution Sequences**

Inserts or deletes DMAP statements by allowing a global “string” search across all subDMAPs within the current solution sequence.

**Format:**

MALTER 'string1'[(occurrence,offset)] , ['string2'[(occurrence,offset)]]

or

MALTER 'string1'[(occurrence,offset)] , [k2]

**Examples:**

1. The following MALTER will insert a MATPRN DMAP statement to print the KJJ matrix for each superelement.

```
SOL 101
MALTER 'MALTER:AFTER SUPERELEMENT STIFFNESS .* GENERATION'
MESSAGE //'/SEID='/SEID $
MATPRN KJJZ/ $
```

2. The following MALTER will add a user DMAP after the PREFACE modules in SOL 100 (USERDMAP).

```
SOL 101
MALTER 'AFTER CALL PREFACE'
.
.
.
```

| Descriptor          | Meaning   |
|---------------------|---|
| 'string1'           | If 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.   |
| 'string1','string2' | DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequence DMAP statements.           |
| k2                  | If k2 is specified, it is applied to the subDMAP in which 'string1' was found (Integer > 0).  |
| occurrence          | This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP (Integer > 0; Default=1).  |
| offset              | This flag indicates the offset from the referenced DMAP statement. Depending on the sign, the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement (Integer; Default = 0). |

**Remarks:**

1. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.
2. The MALTER statement can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
  - K2 or 'string2'(occurrence,offset) references must refer to a DMAP line number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single MALTER statement.
  - 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another MALTER that references the same subDMAP.
3. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement; i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string; all blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained.
4. The special characters used for string searching are described in Remark 9. The characters <, >, and \$, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string

```
IF (DDRMM >=-1)
```

the command is

```
ALTER 'IF (DDRMM \>=-1)' $
```

5. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (""). For example, the alter statement

```
MALTER 'string1' (r1,01), ''
```

is equivalent to

```
MALTER 'string1' (r1,01), 'string1' (r1,01)
```

The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

As another example, the alter statement

```
MALTER 'string1' (r1,01), '' (r2,02)
```

is equivalent to

```
MALTER 'string1' (r1,01), 'string1' (r2,02)
```

6. The existing COMPILE statement options, such as LIST, XREF, SOUIN, etc., cannot be directly specified on the new MALTER statement. They are obtained as follows:

- If a COMPILE statement exists for the subDMAP referenced by the MALTER, then options from this COMPILE statement will be used.

Otherwise, they will be taken from the COMPILER statement, with the exception that the LIST, and SORT option is always on.

7. The MALTER string search order is as follows:
  - All COMPILE statement references that are part of the existing solution sequence (i.e., SOL=) are searched first.
  - Then, all remaining subDMAPs in the solution sequence are searched in ascending alphabetical order.
  - Within a subDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP (not at the current position of the last string match).
8. The MALTER statement must not exceed 72 characters (no continuations are allowed).
9. Metacharacters:
 

|                 |   |
|-----------------|---|
| .               | Matches any <i>single</i> character except <i>newline</i> .   |
| *               | Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since “.” (dot) means any character, “.*” means “match any number of characters.”  |
| [...]<br>or < > | Matches any one of the characters enclosed between the brackets. For example, “[AB]” matches either “A” or “B”. A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example “[A-Z]” will match any uppercase letter from A to Z and “[0-9]” will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character not in the list. |
| ^ or !<br>or .  | Requires that the following regular expression be found at the beginning of the line.   |
| \$              | Requires that the preceding regular expression be found at the end of the line.   |
| \               | Treats the following special character as an ordinary character. For example “\.” stands for a period and “\*” for an asterisk. Also, to search for a tic (‘), the search string must be “\’”.  |
| ,               | Marks the beginning and end of a pattern to be matched.   |
| Note:           | Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ → ! and [ ] → <>) if portability is required. However, all the preceding characters are recognized by MSC Nastran.   |
10. Labels for use with the MALTER have been included in the solution sequences. See [Table 1](#). These labels will be maintained in future versions and it is strongly suggested that alters which use the MALTER command take advantage of the unique MALTER labels. Use of the MALTER labels will significantly reduce the time required to convert alters between versions.

Table 1 DMAP Labels and Corresponding SubDMAP Positions

| DMAP MALTER Labels   |
|--|
| \$MALTER:AFTER PREFACE MODULES   |
| \$MALTER:TOP OF PHASE 1 SUPERELEMENT LOOP, AFTER PARAMETERS AND QUALIFIERS SET   |
| \$MALTER:AFTER SUPERELEMENT STIFFNESS, VISCOUS DAMPING, MASS, AND ELEMENT STRUCTURAL DAMPING GENERATION (KJJZ, BJJZ, MJJZ, K4JJ) |
| \$MALTER:AFTER X2JJ MATRICES READ (K2JJ, M2JJ, B2JJ)   |
| \$MALTER:AFTER TOTAL SUPERELEMENT STIFFNESS, VISCOUS DAMPING, AND MASS FORMULATED, STRUCTURAL + DIRECT INPUT                     |
| \$MALTER:AFTER SUPERELEMENT LOAD GENERATION (PJ)   |
| \$MALTER:AFTER UPSTREAM SUPERELEMENT MATRIX AND LOAD ASSEMBLY (KGG, BGG, MGG, K4GG, PG)  |
| \$MALTER:AFTER SUPERELEMENT MATRIX AND LOAD REDUCTION TO A-SET, STATIC AND DYNAMIC (KAA, KLAA, MAA, MLAA, BAA, K4AA, PA)         |
| \$MALTER:BOTTOM OF PHASE 1 SUPERELEMENT LOOP   |
| \$MALTER:AFTER X2PP MATRICES READ (K2PP, M2PP, B2PP)   |
| \$MALTER:AFTER SUPERELEMENT DISPLACEMENT RECOVERY (UG)   |
| \$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT1 (OUGV1, OES1, OEF1, ETC.)                                       |
| \$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT2 (OUGV2, OES2, OEF2, ETC.)                                       |
| \$MALTER:BOTTOM OF SUPERELEMENT DATA RECOVERY LOOP   |
| \$MALTER:USERDMAP - AFTER CALL PREFACE   |

**MODEL\_CHECK** Specifies Model Check Options

Specifies model checkout run and specifies options to be used.

Format:

$$\text{MODEL_CHECK} \left[ \begin{array}{l} \text{MAT\_DENSITY} = \left\{ \begin{array}{l} \text{OFF} \\ \rho \\ \text{DEFAULT} \end{array} \right\} \\ \left[ \begin{array}{l} \text{MAT\_TECO} = \left\{ \begin{array}{l} \text{OFF} \\ \alpha \\ \text{DEFAULT} \end{array} \right\}, \right. \\ \left. \left[ \begin{array}{l} \text{MAT\_TEIJ} = \left\{ \begin{array}{l} \text{OFF} \\ \alpha_{ij} \\ \text{DEFAULT} \end{array} \right\} \right] \right] \\ \left[ \begin{array}{l} \text{MAT\_DAMPING} = \left\{ \begin{array}{l} \text{OFF} \\ g_e \\ \text{DEFAULT} \end{array} \right\}, \end{array} \right. \end{array} \right]$$

[CHECKOUT][PRINT = item\_list]

Examples:

1. Execute a basic model checkout run. No special output is required.

MODEL\_CHECK CHECKOUT

2. Execute a model checkout run. Print coordinate system and basic grid point data.

MODEL\_CHECK CHECKOUT PRINT=(CSTM,BGPDT)

3. Execute a full solution. Modify the material density temporarily to a value of 0.0.

MODEL\_CHECK MAT\_DENSITY=OFF

or

MODEL\_CHECK MAT\_DENSITY=0.0

4. Execute a full solution. Temporarily modify the values for material density and thermal expansion coefficient.

MODEL\_CHECK MAT\_DENSITY=0.001 MAT\_TECO=1.0 MAT\_TEIJ=0.0

| Descriptor    | Meaning  |
|---------------|--|
| MAT_DENSITY   | Selects material density processing option.                              |
| $\rho$        | Value to be used for the density.  |
| MAT_TECO      | Selects material thermal expansion direct coefficient processing option. |
| $\alpha$      | Value to be used for the thermal expansion direct coefficients.          |
| MAT_TEIJ      | Selects material thermal expansion shear coefficient processing option.  |
| $\alpha_{ij}$ | Value to be used for the thermal expansion shear coefficients.           |
| MAT_DAMPING   | Selects material structural element damping processing option.           |

| Descriptor     | Meaning  |
|----------------|--|
| g <sub>e</sub> | Value to be used for the structural element damping coefficient.   |
| OFF            | Sets material property value to zero.  |
| DEFAULT        | Material property value is set to system default value. See Remark 3.  |
| CHECKOUT       | Selects model checkout solution option. See Remark 5.  |
| PRINT          | Selects items to be printed during model checkout solution.  |
| item_list      | List of model data items to be printed during model checkout run. If more than one item is specified, enclose the list in parenthesis. See Remark 6. |

**Remarks:**

1. The MODEL\_CHECK statement is ignored in RESTART runs.
2. The values specified for material properties using the MODEL\_CHECK statement will be used to temporarily update data for all MAT1, MAT2, MAT3, MAT8, and MAT9 Bulk Data entries only for the duration of the run. These values do not replace data specified on the MAT*i* Bulk Data entries. Caution should be used when postprocessing results via the PARAM POST options since operations using inconsistent data could be performed. Furthermore, when layered composite element properties and materials (PSHELL and MAT2 MID1/MID2/MID3/MID4) are generated, these equivalent MAT2 property entries are not considered to be original input data and the effects of the MODEL\_CHECK directives are permanently reflected in these MAT2 properties. Restarts should not be attempted in this case.
3. System default values of 0.0 have been defined for each of the properties. The defaults can be changed using the following Nastran statement keywords: DEF\_DENS for MAT\_DENSITY, DEF\_TECO for MAT\_TECO, DEF\_TEIJ for MAT\_TEIJ, and DEF\_DAMP for MAT\_DAMPING.
4. The MAT\_TECO descriptor causes the direct components of the thermal expansion coefficient to be modified. The MAT\_TEIJ descriptor causes the shear components of the thermal expansion coefficient to be modified.
5. The CHECKOUT option has the same effect as a PARAM,CHECKOUT,YES Bulk Data entry.
6. The following table summarizes the acceptable specifications for the PRINT item\_list.

| Value | Output Generated            | Parameter |
|-------|-----------------------------|-----------|
| CSTM  | Coordinate systems          | PRTCSTM   |
| BGPDT | Basic grid point data       | PRTBGPDT  |
| GPTT  | Grid point temperature data | PRTGPTT   |
| MGG   | G-set mass matrix           | PRTMGG    |
| PG    | G-set load vectors          | PRTPG     |

See the DMAP parameter descriptions in Section 5 for a discussion of the parameter name in the last column of the table and the output generated. The specification of a print item has the effect of adding a PARAM,parameter,YES entry to the Case Control Section of the file.

**SOL**

## Executes a Solution Sequence

Specifies the solution sequence or main subDMAP to be executed.

**Format:**

```
SOL { n
      subDMAP-name } [SOLIN = obj-DBset NOEXE]
```

**Examples:**

1. In the following example, SOL 103 is executed from MSCOBJ.

```
SOL 103
```

2. In the following example, the PHASE0 subDMAP is altered, SOL 103 is relinked onto the OBJSCR DBset (which is the default for SOLOUT), and SOL 103 is executed.

```
SOL 103
COMPILE PHASE1
ALTER 'DTIIN'
TABPT SETREE,,,// $ 
.
.
.
ENDALTER $
```

3. In the following example, the solution sequence called DYNAMICS is executed from the USROBJ DBset.

```
SOL DYNAMICS SOLIN = USROBJ
```

| Descriptor   | Meaning   |
|--------------|---|
| n            | Solution number. See Remark 6. for the list of valid numbers (Integer > 0).   |
| subDMAP-name | The name of a main subDMAP. See the <a href="#">MSC Nastran DMAP Programmer's Guide</a> (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). |
| obj-DBset    | The character name of a DBset where the OSCAR is stored. See Remarks 1. and 2. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).          |
| NOEXE        | Suppresses execution after compilation and/or linkage of the solution is complete. Also, the Bulk Data Section and Case Control Section are not read or processed.                        |

**Remarks:**

1. If SOLIN keyword is not given and if there are no LINK statements within the input data, the program will perform an automatic link. The program will first collect the objects created in the current run by the COMPILE statement and the remaining objects stored in the MSCOBJ DBset. The program will then perform an automatic link of the collected objects.
2. If the SOLIN keyword is not given but a LINK statement is provided, the SOLIN default will be obtained from the SOLOUT keyword on the LINK statement.
3. The operation sequence control array (OSCAR) defines the problem solution sequence. The OSCAR consists of a sequence of entries with each entry containing all of the information needed to execute one step of the problem solution. The OSCAR is generated from information supplied by the user's entries in the Executive Control Section.
4. The SOLIN keyword will skip the automatic link and execute the OSCAR on the specified DBset.
5. The DOMAINSOLVER may be used in conjunction with solution sequences 101, 103, 108, and 111 to select domain decomposition solution methods.
6. The following Solution Sequences are currently available in MSC Nastran:

**Table 2      Solution Sequences**

| SOL Number | SOL Name | Description  |
|------------|----------|--|
| 101        | SESTATIC | Statics with options:<br>Linear steady state heat transfer.<br>Alternate reduction.<br>Inertia relief. |
| 103        | SEMODES  | Normal modes.  |
| 105        | SEBUCKL  | Buckling with options:<br>Static analysis.<br>Alternate reduction.<br>Inertia relief.                  |
| 106        | NLSTATIC | Nonlinear or linear statics.   |
| 107        | SEDCEIG  | Direct complex eigenvalues.  |
| 108        | SEDFREQ  | Direct frequency response.   |
| 109        | SEDTRAN  | Direct transient response.   |
| 110        | SEMCEIG  | Modal complex eigenvalues.   |
| 111        | SEMFREQ  | Modal frequency response.  |
| 112        | SEMTRAN  | Modal transient response.  |
| 114        | CYCSTATX | Cyclic statics with option:<br>Alternate reduction.  |
| 115        | CYCMODE  | Cyclic normal modes.   |
| 116        | CYCBUCKL | Cyclic buckling.   |

Table 2 Solution Sequences

| SOL Number | SOL Name  | Description   |
|------------|---|---|
| 118        | CYCFREQ   | Cyclic direct frequency response.   |
| 128        | SENLHARM  | Nonlinear Harmonic Response   |
| 129        | NLTRAN  | Nonlinear or linear transient response.   |
| 144        | AESTAT  | Static aeroelastic response.  |
| 145        | SEFLUTTR  | Aerodynamic flutter.  |
| 146        | SEAERO  | Aeroelastic response.   |
| 153        | NLSCSH  | Static structural and/or steady state heat Transfer analysis with options:<br>Linear or nonlinear analysis.   |
| 159        | NLTCSH  | Transient structural and/or transient heat Transfer analysis with options:<br>Linear or nonlinear analysis.   |
| 200        | DESOPT  | Design optimization.  |
| 400        | NONLIN  | Nonlinear Static and Implicit Transient Analysis and all linear sequences from statics, modes, frequency, and transient inclusive with perturbation analysis based on previous nonlinear analysis |
| 600        | SESTATIC ( See <a href="#">SOL 600, ID, 153</a> ) | MSC Nastran API into MSC MARC nonlinear   |
| 700        | NLTRAN ( See <a href="#">SOL 700, ID, 172</a> )   | Nonlinear Explicit Transient Analysis   |

## SOL 600, ID

Executes Marc from Inside Nastran

Creates Marc input and optionally executes Marc from inside SOL 600.

**Format:**

```
SOL 600, ID PATH= COPYR= NOERROR MARCEXE=SOLVE NOEXIT
OUTR=op2,xdb,pch,f06,eig,dmap,beam, sdrc,pst,cdb=(0, 1, 2, or 3) STOP= CONTINUE=
S67OPT= MSGMESH= SCRATCH= TSOLVE= SMEAR PREMGLUE MRENUELE= MRENUGRD=
MRENUMBR= SYSabc= S6NEWS=
```

**Examples:**

```
SOL 600,106 STOP=1
SOL 600,106 OUTR=OP2,F06
SOL 600,106 PATH=/progs/marc2003/tools OUTR=op2,f06
SOL 600,129 PATH=1 STOP=1
SOL 600,129 PATH=1 OUTR=OP2,CDB=0
SOL 600,106 PATH=1 CONTINUE=1
SOL 600,106 PATH=1 MARCEXE=SOLVE OUTR=OP2
SOL 600,153 PATH=1 STOP=1 TSOLVE=M
SOL 600,106 OUTR=OP2,F06 SMEAR
SOL 600,106 OUTR=OP2 PERMGLUE
SOL 600,106 PATH=1 STOP=1 MRENUELE=2
SOL 600,106 PATH=1 STOP=1 MRENUGRDD=2
SOL 600,106 PATH=1 STOP=1 MRENUMBR=2
SOL 600,106 STOP=1 SYS001=8193 SYS9=6000
SOL 600,106 STOP=1 S6NEWS=YES
```

SOL 600, ID is an Executive Control statement similar to SOL. The difference between SOL and SOL 600, ID is that the computations (element matrix formulations, matrix decomposition, etc.) will be performed by Marc rather than by Nastran. Inputs and outputs as much as possible will be the same as (or similar to) the familiar Nastran inputs and outputs, however standard Marc inputs and outputs are also available. SOL 600 is primarily intended for nonlinear static and dynamic analysis of 3D structures that have already been manufactured and assembled. Although it has capabilities for 2D structures and for certain manufacturing processes, those capabilities should only be used for “simple” cases. For more complex 2D and manufacturing analyses either a standalone version of Marc or SOL 400 is recommended.

The SOL 600, ID statement should normally be used only for nonlinear analysis, but it may also be used for certain classes of linear static or dynamic analyses. The recommended form of this command is shown with the options provided above. If entered with “SOL 600, ID” only, it acts just like SOL except a Marc input data file “jid.marc.dat” will be generated (“jid” is the name of the Nastran input file without the extension.) For example, if the Nastran input file is named abcd.dat, (or abcd.bdf) then “jid”=abcd.

The required ID represents many valid solution sequence integer or names shown in [Table 2](#) for the SOL statement. Examples are 106, 129, NLSTATIC, NLTRAN. The following solutions are not available: 107, 108, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 190, 200, and 400 (and their equivalent names). Solutions specified in [Table 2](#) of the SOL statement may be used. If the model has contact, ID must be 106, 129, 153, 159 or their equivalent names unless PERMGLUE is used.

Although SOL 600 supports 2D analyses (axisymmetric and plane strain), the support is not complete. It is strongly recommended that 2D analyses use some other solution sequence.

Most items on the SOL 600, ID after ID itself may be specified in environmental variables. This may be done any way environmental variables can be set. They may be set by the Nastran user at run time or by the system administrator when Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the [MSC Nastran 2020 Installation and Operations Guide](#). A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user's rc file, a local rc file or in a script used to submit Nastran. Any string or value listed on the SOL 600, ID statement is also valid as an environmental variable. If the environmental variables are placed in the system-wide rc file, they may be used by a company for all Nastran users and even hide the fact that Marc is being spawned if so desired.

The following environmental variables are available:

| Environmental Variable | Item on SOL Statement                          |
|------------------------|--|
| NASM_PATH              | PATH   |
| NASM_COPYR             | COPYR  |
| NASM_OUTR              | OUTR   |
| NASM_STOP              | STOP   |
| NASM_NOERROR           | NOERROR  |
| NASM_STRFILE           | Path and name of marcfilt.txt file (see below) |

## PATH

PATH is an optional item which determines the location of the version of Marc to be executed. If PATH is omitted, the version of Marc included with Nastran will be used if it can be located. In this case, the run script for Marc (run\_marc or run\_marc.bat) will be expected to be in a directory under /MSC\_BASE. MSC\_BASE is an environmental variable set when Nastran first starts execution that defines the base installation directory for Nastran. If for some reason MSC\_BASE cannot be determined, the commands to spawn Marc will fail and the user must re-run Nastran with one of the PATH options set or the NASM\_PATH environmental option set to the desired location of Marc's tools directory.

## PATH=1

If PATH=1 is specified, Nastran will determine the proper command to execute the companion program. To aid Nastran in determining the program's location, a file named marcrun.pth must be available in the same directory where the Nastran input file resides. The marcrun.pth file must contain one line providing the location (complete path) of the run\_marc script. A typical example of the line in the file marcrun.pth would be

```
/mycomputer/marc200x/tools
```

To this path is appended the string “/run\_marc -jid name.marc -v no” and possibly other items to form the complete string used to execute Marc. This complete string looks like the string shown in the following

PATH=3 example. Note that on Windows systems, substitute a back slash for the forward slashes shown. Do not terminate the line with a forward slash or back slash.

## PATH=2

If PATH=2 is specified, it is expected that the directory with the run\_marc script is on the PATH. If PATH=2 is specified, Marc will be executed from inside Nastran using the command:

```
run_marc -jid jid.marc.dat -v no
```

## PATH=3

When PATH=3 is specified, the complete command to execute Marc must be contained in a file named marc.pth (lowercase). This file should typically contain one line of the form:

```
/mycomputer/marc200x/tools/run_marc -jid name.marc -v no
```

## COPYR

COPYR is an optional item. If COPYR is specified, Marc output files will be copied to Nastran output files and/or deleted according to the options shown in the following table:

| COPYR Option        | Copy Marc Output Files to Nastran Output Files | Delete Marc Input & Output Files |
|---------------------|--|----------------------------------|
| 0 (default)         | No   | No                               |
| 1 or -1 (see below) | Yes  | Yes                              |
| 2 or -2 (see below) | Yes  | No                               |
| 3                   | No   | Yes                              |

If COPYR is 1 or 2, Marc's out and log files will be copied exactly as produced by Marc.

If COPYR is -1 or -2 the actions as shown above for +1 or +2 will occur, and Marc-type text will be converted to Nastran-type text (or any other desired text) using an ASCII file named marcfilt.txt. This file must be located in the same directory where the Nastran input resides or in the same directory where the Marc executable resides. The marcfilt.txt file can contain as many lines as desired like the one shown below:

```
"Marc string 1" "Replacement String 1"
"Marc string 2" "Replacement String 2"
```

That is, each line contains two strings. Each string starts and ends with a double quote sign (""). The Marc string must match the exact content and case as found in the Marc .out or .log files. The replacement string may be any string desired and can be the same length, shorter or longer than the Marc string. The two strings must be separated by at least one space, but more spaces are acceptable. Line lengths for marcfilt.txt, as well as Marc's .out and .log files are limited to 200 characters for the text replacement option.

The following Marc files are potentially affected by the COPYR option:

| Marc Output File           | Nastran Output Copied to            | COPYR        |
|----------------------------|-------------------------------------|--------------|
| name.marc.out              | name.f06                            | 1, 2, -1, -2 |
| name.marc.log              | name.log                            | 1, 2, -1, -2 |
| name.marc.t16              | Not copied, will remain if produced |              |
| name.op2, fort.11 or ftn11 | Not copied, will remain if produced |              |

### MARCEXE=SOLVE

MARCEXE=SOLVE is an optional item. If MARCEXE is entered, an existing input file named jid.marc.dat is assumed to exist in the directory where the run was submitted. Nastran will execute Marc using the existing jid.marc.dat file. A new Marc file will not be created. Other options available when MARCEXE is used are PATH and OUTR. Options not available with MARCEXE are COPYR, STOP, NOEXIT, NOERROR and CONTINUE. Beware that the original jid.marc.dat will be renamed to jid.marc.dat.1 automatically by Nastran just like an existing jid.f06 is renamed to jid.f06.1

### NOERROR

NOERROR is an optional item. If NOERROR is specified, errors due to features that are available in Nastran but not available in Marc, and/or features not yet supported by the translator will be ignored (see Restrictions and Limitations). If NOERROR is entered and STOP=2 (or 3) is not specified, Marc will be executed even though the complete Nastran model may not have been completely translated. We recommend that NOERROR only be used by experienced analysts and then only with extreme caution.

### MSGMESH

MSGMESH is an optional item and should be omitted unless the Bulk Data contains some entries in the MSGMESH format. If there are any MSGMESH entries present, set MSGMESH=Nhigh, where Nhigh is an integer equal to the largest node, element, material or property in the model (after MSGMESH entries have been expanded). Optionally this item can be set as an environmental variable S600\_M001=Nhigh. If MSGMESH entries are present and neither S600\_M001 nor MSGMESH on the SOL 600 statement are set, the job will terminate with an appropriate message.

### NOEXIT

NOEXIT is an optional item. If entered, the DMAP generated “on the fly” to process the OUTR options will not contain EXIT and Nastran will proceed. This means in most cases, the Nastran solution as well as the Marc solution will occur. If .f06 is specified as one of the OUTR options, this could cause confusing output as both the Marc and Nastran results will be in the .f06 file. Confusion could also result from both outputs being in .op2, .xdb and/or .pch files. Therefore, this option should only be used with great care.

Listing of the DMAP generated on the fly for SOL 600 can be suppressed by placing ECHOOFF just after the SOL 600 entry.

**Restrictions:**

- OUTR options may not be used with restart jobs.
- OUTR options are not available for SOL 600 2D analysis such as axisymmetric or plane strain.
- OUTR options are not available for SOL 600 heat transfer.
- Spaces in the list of OUTR items are not allowed.

**OUTR**

OUTR is an optional item. If OUTR is specified, Marc output results will be converted to various types of Nastran formats. The OUT option on the Nastran command should not be used with any OUTR options. The type of output to be produced depends on the OUTR options entered as well as any DMAP entered in the executive control. If OUTR is omitted, no Marc output will be brought back into Nastran, but standard Marc .out, .t16 and/or .t19 as well as an op2 file will be available depending on the options selected with PARAM,MARCT16, PARAM,MARCT19 and other options. OUTR options may not be used with restart jobs.

The following options are available:

**Option 1 -- Specify a String of Desired Output Types (Preferred Method)**

OUTR=OP2,F11,F06,PCH,XDB,T16,T19,PST

Use any or all of the above to request the following options except that both OP2 and XDB should not be entered:

|      |  |
|------|--|
| OP2  | Create output2 file named jid.op2 consisting of input model and output results datablocks. This option requires PARAM,POST,-1 or PARAM,POST,-2 in the Bulk Data.   |
| F11  | Create output file fort.11 or ftn11 (depending on the computer system) consisting of output results datablocks only.   |
| F06  | Put Marc output results (displacements, stresses, strains) in Nastran's jid.f06 file using OFP. The resulting output will look just like any standard Nastran run.   |
| PCH  | Create punch file named jid.pch with Marc's output in standard Nastran punch format.   |
| XDB  | Create .xdb database file named jid.xdb with input model and output results. This option requires PARAM,POST,0 in the Bulk Data. XDB is not available with the eig option and if entered will switch to the OP2 option.  |
| eig  | The eig option must be specified if .op2, .xdb, .pch, or .f06 options are specified and Marc performs natural frequency or buckling eigenvalue analysis. The reason it must be provided on the SOL entry is to enable Nastran to create DMAP on the fly which include the LAMA data block. If the eig option is omitted, eigenvectors will be present in the Nastran output but no eigenvalues will be available. The beam and eig options are mutually exclusive (you cannot specify both). |
| BEAM | The BEAM option must be specified if .op2, .xdb, .pch, or .f06 options are specified and you want to place internal loads in any of these files. The BEAM option is not available for Windows systems.   |

- SDRC      An SDRC op2 file will be produced. PARAM,POST,-2 is also necessary in the Bulk Data for this option. Note: The datablocks might be in a different order than for other solution sequences.
- T16      Marc's results will be saved during the Marc execution on a binary (or unformatted) file named jid.marc.t16 (this happens by default and does not need to be specified on the SOL 600 line).
- T19      Marc's results will be saved during the Marc execution on an ASCII file named jid.marc.t19. The t19 file will normally be saved if param,maract19,1 is entered.
- PST      Nastran will be run to output a previous Marc run's results contained on t16 file in the desired forms (OP2,F11,F06,PCH and/or XDB). The appropriate OUTR T16 options must be selected in addition to PST (specify one or more of OP2,F11,F06,PCH and/or XDB.) Nastran will not be run past IFP and is used only to perform the desired output results conversions. A previous Marc t16 file must be copied to the new jid.marc.t16.1 (you may not process XDB and OP2 in the same run.)
- DMAP      The user will enter his own DMAP to create whatever type of output that is desired, such as .op2, .xdb, .pch, or .f06. For all other options, DMAP as needed is generated internally by Nastran.
- cdb      3D Contact will be output in one of the datablocks described below:
- 0      Store output in OESNLBR and OESNLXR (OESNLXR will be empty like SOL 106)
  - 1      Store output in OESNLBR
  - 2      Store output in OESNLBD
  - 3      Store output in OESNLXD
- Note:**
1. SOL 106 outputs both OESNLBR and OESNLXR but OESNLXR is empty.
  2. SOL 129 only outputs the OESNLXD datablock and it is empty.
  3. Case Control BOUTPUT is also required to obtain this type of output.
  4. The default is 0 if contact is present in the model and OUTR=op2 (or .xdb, punch and/or .f06).
  5. This option is specified like the example shown:  
OUTR=OP2,F06,CDB=0
  6. The datablocks have the same names and type of information whether executing SOL 600,106 or SOL 600,129.

### Option 2 -- Specify an Integer to Select Certain Options (Not Recommended)

OUTR=1 or 2 and an op2 file named fort.11 or ftn11 will be produced and DMAP as shown below is required to bring the Marc output results back into the Nastran database.

```
COMPILE NLSTATIC
ALTER 'SUPER1' $
INPUTT2 /OUGV1,OES1,OSTR1,TOL,/-1/11 $
OFP OUGV1,OES1,OSTR1//0/1 $
EXIT $
```



The 1 at the end of the OFP statement produces output in the .f06 file. If a punch file is also needed, change the 1 to a 5. If an XDB file is also needed, add the following lines just after the OFP line:

```
DBC TOL,CASECC,'OL'/'CASECC'-1/DBCPATH/S,N,CP//TRAN//GEOMU/LOADU/POSTU/
DBCDIAG/DBCCONV/DBCOVWRT $  

DBC OUGV1,OES1,'OUG'/'OES'-1/DBCPATH/S,N,CP//TRAN//GEOMU/LOADU/POSTU/
DBCDIAG/DBCCONV/DBCOVWRT $  

DBC OSTR1,'OES'-1/DBCPATH/S,N,CP//TRAN//GEOMU/LOADU/POSTU/
DBCDIAG/DBCCONV/DBCOVWRT $
```

OUTR can be set to one of the following values to automatically produce the output in Nastran form without entering any DMAP. In fact, no DMAP should be entered for the options greater than 2 shown:

Table 4-3 Integer Options Available Using SOL 600 OUTR Option --  
Nastran Output Results Produced When Marc Exits

| OUTR (IO) | OP2 with Input Datablocks | fort.11 or ftn11 Output Datablocks Only | .f06 (Print) | .pch (Punch) | .xdb | Marc File Used |
|-----------|---------------------------|---|--------------|--------------|------|----------------|
| 1         | N                         | Y                                       | N            | N            | N    | .t19           |
| 2         | N                         | Y                                       | N            | N            | N    | .t16           |
| 16        | Y                         | Y                                       | N            | N            | Y    | .t16           |
| 166       | Y                         | Y                                       | Y            | N            | Y    | .t16           |
| 266       | Y                         | Y                                       | N            | Y            | Y    | .t16           |
| 366       | Y                         | Y                                       | Y            | Y            | Y    | .t16           |
| 19        | Y                         | Y                                       | N            | N            | Y    | .t19           |
| 199       | Y                         | Y                                       | Y            | N            | Y    | .t19           |
| 299       | Y                         | Y                                       | N            | Y            | Y    | .t19           |
| 399       | Y                         | Y                                       | Y            | Y            | Y    | .t19           |

If OUTR = -1, -2, -16, -166, -266, -366, -19, -199, -299 or -399 only the output conversion process takes place. An Marc input file is not produced, Marc is not spawned from Nastran, but .op2, .xdb, .pch and/or .f06 results can be produced. For such cases, the Case Control and Bulk Data files can be dummies (for example, they can contain several nodes and one element) or a full file could be used. These options are handy if Marc is run by modifying the Marc input file (jid.marc.dat) with an editor or for someone who creates Marc input and runs Marc outside the Nastran environment, but wants output in one of the Nastran formats (see Remark 6).

## STOP

STOP is an optional item. STOP is used to prevent execution of Marc or exit Nastran after IFP, if so desired. DO NOT ENTER any of the STOP options if any of the OUTR options are entered as the DMAP generated automatically by Nastran will put an EXIT in the proper place. The various options are as follows:

### STOP=0

If STOP=0 Nastran will not be stopped after Marc exits. Nastran will attempt to obtain its own solution to the problem if possible. Use of this option can lead to confusion because results from both Marc and Nastran will be available. If the Marc results are placed in the .f06 file and if the Nastran results are also available in the .f06 file, it will be difficult to tell which results came from Nastran and which results came from Marc. This also applies to .op2 files and .xdb files. It is suggested the STOP=0 option be used by extremely experienced SOL 600 users and even then with great care.

### STOP=1

If STOP=1 Nastran will be gracefully stopped after IFP. This option is used to prevent Nastran from performing its own solution (normally used when the solution is performed by the Marc). STOP=1 should be normally used if OUTR is not specified. STOP=1 is the default if no STOP, CONTINUE or OUTR options are entered.

### STOP=2

For STOP=2 Marc will not be executed. This option is used if you wish to examine the Marc input file and make changes prior to running Marc. However, if STOP=2 is entered, the OUTR options will not be available.

### STOP=3

STOP=3 is a combination of STOP=1 and STOP=2. Nastran is stopped after IFP and Marc is not executed. This would be the normal STOP option if you want to examine a Marc input file, then execute Marc manually. The STOP=2 option is normally used if you want to obtain comparative results between standard Nastran solutions and Marc solutions (in which case, all input options must be fully supported by both programs). If STOP=3 is entered, the OUTR options will not be available.

### CONTINUE=

CONTINUE= specifies an option as to how Nastran will continue its analysis after Marc finishes. For this to happen, do not enter any STOP or OUTR options. It is not usually possible to perform more than one of these operations if necessary.

- 0 Nastran will continue the current solution sequence as normal. For example if SOL 600,106 is entered, SOL 106 will continue as normal after Marc finishes. Of course, no 3D contact or materials not supported by SOL 106 may be used.
- 1 Nastran will switch to SOL 107 to compute complex eigenvalues. Marc will generate DMIG matrices for friction stiffness (and possibly damping) on a file specified by pram,marcfil1,name and time specified by param,marcstif,time. This is accomplished by making a complete copy of the original Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file. The user must put CMETHOD and CEIG in the original Nastran input file.
- 2 (Option not presently available.) Nastran will switch to SOL 107 to compute complex eigenvalues. Marc will generate OUTPUT4 matrices for friction stiffness (and possibly damping) on a file specified by pram,marcfil2,name and time specified by param,marcstif,time. This is accomplished by making a complete copy of the original Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file.
- The original Nastran file should include CMETHOD=id in the Case Control command and a matching EIGC entry in the Bulk Data.  
In addition, the DMIG entries specified by MDMIOUT will be included in a separate Nastran execution spawned from the original execution. Case Control and Bulk Data will be added to the original input to properly handle these matrices in the spawned Nastran execution.
- 6 Same as option 1 except SOL 110 is run. For this option, the original Nastran input file must contain METHOD=ID1 and CMETHOD=ID2 in the Case Control as well as matching EIGRL (or EIGR) and EIGC entries in the Bulk Data.
- 7 Same as option 1 except SOL 103 is run for real eigenvalues/eigenvectors. The database can be saved to restart into SOL 110 if desired. This should be done on the command line or in a rc file with scratch=no. For this situation, the original Nastran input file must include METHOD=id in the Case Control command and a matching EIGRL or EIGR entry in the Bulk Data. (CMETHOD and EIGC can also be included.) The actual restart from SOL 103 to 110 must be performed manually at the present time.
- 101+ Continue options 101 to 400 are used to convert Marc's initial contact tying constraints to MPC's and then continue in SOL 101 to 112 as a standard Nastran execution. For example, if CONTINUE=101, a SOL 101 run with all the geometry load cases, etc. from the original run would be conducted with the addition of the initial contact MPC determined from Marc. The continue=101+ options are frequency used to model dissimilar meshes as well as glued contact which does not change throughout the analysis. This option can be used for any standard Nastran sequence where the initial contact condition does not change. In order for initial contact to work, the surfaces must be initially touching. If they are separated by a gap, the MPC's will be zero until the gap closes and thus the initial MPC's are zero. This option automatically sets BCPARA INITCON=1.

## S67OPT=NO

If S67OPT=NO is entered the following action will be taken for SOL 600 or SOL 700: TA1MCK and EMGPRO will not be disabled (when these routines are disabled, materials used only by SOL 600 or SOL 700 such as MATG, MATF, MATHP, etc. may be in the model and the t16op2 conversion will take place, otherwise the job will fail with a FATAL ERROR). Also, Case Control FATAL error termination will occur at the same place as other Nastran Solution Sequences. If S67OPT=YES or S67OPT is omitted entirely, TA1MCK and EMGPRO will be disabled and Case Control FATAL ERRORS will cause job termination immediately. S67OPT=YES is the default.

## SCRATCH=

Determines what will happen when a SOL 600 job is initiated with SOL 600 database files (\*.3dc, \*.prp) present in the run directory (this usually means another job is running and conflicts can occur.) The default is SCRATCH=WAIT01. Options are as follows:

|                |  |
|----------------|--|
| SCRATCH=DELETE | Attempt to delete all *.3dc and *.prp files, continue with present job. If the attempt to delete them fails, the job will terminate with an appropriate message.   |
| SCRATCH=ABORT  | If any *.3dc or *.prp files are found, abort the present job with an appropriate message.  |
| SCRATCH=WAIT   | If any *.3dc or *.prp files are found, wait until they disappear, then begin current run. This option will wait an "indefinite" amount of time.  |
| SCRATCH=WAITxx | If any *.3dc or *.prp files are found, wait for xx minutes or until they disappear, then begin current job. If they do not disappear within xx minutes, abort the current job with an appropriate message. Examples, to wait up to 1 minute, enter SCRATCH=WAIT01, to wait up to 15 minutes, enter SCRATCH=WAIT15. Note: xx can range from 01 to 99. |

## SCRATCH= Remarks:

1. For the WAIT options, if no \*.3dc or \*.prp files are found, the job will start immediately.
2. No spaces are allowed.

## TSOLVE

Determines which "solver" (Nastran or Marc) is used to solve a heat transfer analysis. The default is Nastran.

|           |   |
|-----------|---|
| TSOLVE=M  | Marc is used as the thermal solver.   |
| TSOLVE=MS | Marc is used as the thermal solver followed by a structural analysis using the temperatures from the end of thermal analysis. |
| TSOLVE=N  | Nastran is used as the thermal solver (default).  |

## TSOLVE Remarks:

1. If the default is used and thermal contact is present, SOL 600 spawns Marc to calculate initial thermal contact variables which are then read by Nastran, turned into Nastran CELAS and other variables. A second Nastran run is spawned for the primary Nastran run to complete the heat transfer calculations.
2. If OUT or OUTDIR are used with thermal contact in SOL 600 they must both reference the same directory.
3. This option should be entered for heat transfer analysis only.

## SMEAR

The term SMEAR, as used by SOL 600, is different than that used on the PCOMP Bulk Data entry. For SOL 600, SMEAR is the same as LAM=BLANK on the PCOMP entry. Other LAM options are not available using SOL 600, however complete integration and fast integration methods are available, see the PCOMPF Bulk Data entry.

If the string SMEAR is entered on the SOL 600, ID command line, composite shell entries using PCOMP will use the smeared approach. If SMEAR is not entered, the through-the-thickness integration approach will be used. The smeared approach is identical to other Nastran solution sequences where PCOMP entries are converted to PSHEL and MAT2 entries. The through-the-thickness integration approach is more accurate for post-buckling and nonlinear analyses but takes more computer time. OP2.f06 and punch outputs are available and are controlled by the OUTR options OUTR=xxx where xxx is .op2, .f06 and/or .pch. If any OUTR options are specified, .op2 must be included. In addition, standard Case Control requests are required.

## SMEAR Option Restrictions

1. The SMEAR option may only be used if all composite materials in the model are made of shell elements (if there are any composite solid elements, this option may not be used.)
2. Case Control requests for DISP(options)=ALL, the STRESS(options)=ALL must be entered. STRAIN(options)=ALL is optional. (options) consist of any combination of (print,plot,punch)
3. The SMEAR output options may not be controlled using sets.
4. It is suggested that the Marc t16 file be limited to only those output "items" absolutely necessary as composite output can be large and take significant computer time.
5. If OUT or OUTDIR are used with this option, they must reference the same directory.

## PERMGLUE

Specify PERMGLUE if permanent glued contact is to be used. Permanent glued contact is glued contact where the glued condition is determined using initial contact. This glued condition will remain throughout the analysis. The MPC's produced by the PERMGLUE option are identical to those formed in SOL 101 or SOL 103 when the permanent glue option is specified. When this option is used, set BCONTACT=ALLGLUP. For SOL 600, the PERMGLUE option is the only way contact can be used with SOL600,101 or SOL 600,103 or other "linear" analyses.

## MRENUELE

Determines if SOL 600 elements will be renumbered or not. (Default = 0)

- 0      No renumbering will occur (suggested for models with largest element number less than approximately 20000)
- 1      All elements will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file elenum.txt
- 2      All elements will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

Remarks concerning MRENUELE:

1. MRENUELE must be set on the SOL 600 entry if the maximum element number is greater than 9,999,999.
2. The “=” and an integer of “0” “1” or “2” must follow “MRENUELE” with no spaces when MRENUELE is entered on the SOL 600 entry.
3. If the maximum element is 9,999,999 or smaller MRENUELE may be set as a parameter in the bulk data or placed in a rc file.
4. MRENUELE should not be set on the SOL 600 and as a parameter.
5. For MRENUELE=1 an equivalence list of original and re-numbered element numbers is output on file elenum.txt

## MRENUGRD

Determines if SOL 600 grid ID's will be renumbered or not. (Default = 0)

- 0      No renumbering will occur (suggested for models with largest grid ID less than approximately 20000)
- 1      All grid ID's will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt
- 2      All grid ID's will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

Remarks concerning MREUGRD:

1. MRENUGRD must be set on the SOL 600 entry if the maximum element number is greater than 9,999,999.
2. The “=” and an integer of “0” “1” or “2” must follow “MRENUELE” with no spaces when MRENUGRD is entered on the SOL 600 entry.
3. If the maximum grid ID is 9,999,999 or smaller MRENUGRD may be set as a parameter in the bulk data or placed in a rc file.
4. MRENUGRD should not be set on the SOL 600 and as a parameter.

5. For MRENUGRD=1 an equivalence list of original and re-numbered grid id's is output on file gridnum.txt

### MRENUMBR

Determines both grid and element ID's for SOL 600 will be renumbered or not. (Default = 0)

- 0      No renumbering will occur (suggested for models with largest grid ID less than approximately 20000)
- 1      All grid and element ID's will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt
- 2      All grid and element ID's will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

Remark concerning MREUMBR: All remarks for MRENUELE and MRENUGRD apply.

### S6news

S6news can be set to YES or NO. If set to YES, a list of new features currently considered to be in beta tests are output complete with the Case Control and/or Bulk Data descriptions anticipated once these features are released.

### SYSabc

Allows Nastran system cells to be set on the SOL 600 Executive Control statement if desired. Abc must be an integer ranging from 1 to 500 and be a valid system number. INTT must be an integer with 1-8 digits. More than one SYSabc=INTT statement may be made if so desired, however abc may not be repeated. Note that the limit on the integer following the equal sign is 8 digits.

### Running SOL 600 in Steps -- Modification of the SOL 600 Statement Using Environmental Variables:

It is possible to run the main portions of SOL 600 in single steps without changing the Nastran input file. This is accomplished using one of the two environmental variables discussed below. A user can set these variables in a script that runs Nastran, from the command line or for Windows using the control panel. Note that on Linux systems, the name of the environmental variable must be in upper case. The string to which it is set can be in upper or lower case and will be converted to upper case.

### To Run SOL 600 in Three Steps Without Changing the SOL 600 Statement in the Input File:

First, make sure that your SOL 600 input file has a SOL 600 statement that contains all of the features you would want if all steps were done in a single run. For example, if you wish to make an op2 file and place the results in the .f06 file, a typical SOL 600 statement would be as follows:

SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06

or

SOL 600,NLSTATIC OUTR=OP2,F06

(if the default path to Marc is to be used).

It is important to have the OUTR options specified at the end of the SOL 600 statement. The following environmental variable can be set as shown to run the three steps (a Linux shell example is shown):

1. export MARC\_RUN="stop"

This will tell Nastran to run the internal Nastran-to-Marc translator only. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06 STOP=3
```

This change will be shown in the .f06 file.

2. export MARC\_RUN="solv"

This will tell Nastran to run Marc from inside Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06 MARCEXE=SOLVE
```

For this run, it is assumed that a file named jid.marc.dat resides in the input file directory created from a previous translator-only run. The Nastran script will automatically rename jid.marc.dat to jid.marc.dat.1, but the Nastran executive processing will name it back to jid.marc.dat

3. export MARC\_RUN="pst" This will tell Nastran to run the t16 to op2 translator inside Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06,PST
```

For this run, it is assumed that a file named jid.marc.t16 as well as the original Nastran input file jid.dat resides in the input file. The "t16" file should have been created from a previous Marc execution using the same computer system (cross-platform support is not available for this step). The Nastran script will automatically rename jid.marc.t16 to jid.marc.t16.1, and the Nastran t16op2 conversion routines will look for files with names jid.marc.t16, jid.marc.t16.1, jid.marc.t16.2 up to jid.marc.t16.5 in that order. If no such files are found, the t16op2 job will exit with a message.

### Method to Completely Modify the SOL 600 Statement:

For maximum versatility without having to modify the Nastran input file, the SOL 600 statement can be modified completely using the environmental variable SOL600\_CMD. Assuming that the original SOL 600 statement in jid.dat contains the string:

```
SOL 600,NLSTATIC OUTR=OP2,F06
```

and the environmental variable is set as follows:

```
export SOL600_CMD="SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE"
```

the new SOL command line internal to Nastran will be

```
SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE
```

and Nastran will stop after creating the Marc input file. This would be the same as if the following SOL 600 statement was entered:

```
SOL 600,NLSTATIC PATH=1 STOP=3
```

Any valid SOL 600 statement can be issued using the SOL600\_CMD environmental variable without changing the original Nastran input file at all.

### Remarks:

1. Only one SOL 600, ID job may be run in a directory at any given time. However, if a previous run was made and output files such as name.marc.t16 were produced, they will be renamed name.marc.t16.1, etc. following the Nastran re-naming convention.
2. If OUTR is specified, STOP must not be specified.
3. The COPYR option can be used to delete all files directly created by Marc if the output desired are Nastran files only.
4. When OUTR is specified, the Marc files such as jid.marc.out, jid.marc.t16 will be renamed to jid.marc.out.1 and jid.marc.t16.1 at the start of the run. This renaming is accounted for when opening the files.
5. To generate .xdb files, PARAM,POST,0 must be included in the Bulk Data Section. To generate OP2 files with geometry, PARAM,POST,-1 (for Patran and Femap) or PARAM,POST,-2 (for SDRC) should be included in the Bulk Data.
6. Although SOL 600, ID supports linear analysis (ID=101, 103, 105), not all features are available. For example, Case Control commands, STATSUB, SUBCOM, SUBSEQ, SYMCOM, AUXMODEL, AXISYMMETRIC, CLOAD, DEFORM, HARMONICS, MFLUID, NSM, and REPCASE are not available. For nonlinear analyses (ID=106, 129) Case Control commands, NNLOAD and NONLINEAR are not available.
7. To output displacements in the jid.marc.out file, do the following:  
In the Case Control, set DISP(PRINT)=ALL or DISP(PRINT,PLOT)=ALL  
In the Bulk Data, include the following two parameters:

PARAM,MARCPRNR,1  
PARAM,MARCND99,-1

8. All SOL 600 character variable parameters, such as MRAFFLOW, must be left justified in the starting in field 3.
9. If generation options such as EGRID, GRIDG, CGEN, etc., are used, the MSGMESH option or the environmental variable, S600\_M001 must be set with a value equal to the largest grid ID or element ID (whichever is larger). This is needed for initial SOL 600 memory sizing, which happens before these generation entries are converted to standard Nastran GRID, CQUAD4, etc. entries. SOL 600 can not read the generation features. This environmental variable needs to be included in the script or batch file that runs SOL 600 jobs (for Windows, it can be set in the environment).
10. Fixed load stepping (or time stepping) is controlled primarily by PARAM,MARCITER and PARAM,MARCAUTO rather than NLPARM or NLAUTO.
11. 2D and 3D contact and elements may not be mixed in the same model.
12. For the OUTR options, the only stress tensor available is Cauchy stress (E341). If some other stress tensor is selected using MARCOUT, and E341 is not selected, no stresses will be available in the .op2, .xdb, .pch, or .f06 files.

13. For multi-layer composites, stresses in the preferred direction (E391), also known as the layer direction, are usually necessary. The default Cauchy stresses (E341) will be automatically charged to E391 for composites.
14. SOL 600 supports a new field on RFORCE (2nd line 4th field) to allow different portions of the structure to have different rotation accelerations.
15. 2D Plain stress is available and this type of analysis is achieved by setting MID=-1 on all PSHELL entries and adding the following parameters to the bulk data.

```
PARAM,MRALIAS,011003
PARAM,MALIAS02,006003
```

SOL 600 will use the alias commands to convert plain strain elements that normally are introduced with MID2=-1 into the input into plain stress. Although MALIAS02 specifies that Marc element 3 be used for CTRIAi, the new Marc element 201 plain stress triangle will be used instead of a degenerate element 6 plain stress quad. By default, the CTRIAi node numbers will not be reversed, however if the user wants to reverse them enter PARAM,MREVPLST,1 in the bulk data. Any contact entered into the file needs to specify 2D on all BCBODY entries. 2D plain stress analysis for SOL 600 is available through Patran which will generate the above alias parameters automatically.

### Restrictions and Limitations of SOL 600

Certain features are available in Nastran that are not available in Marc, and vice versa, the following restrictions/limitations are imposed on Nastran. Those restrictions indicated by (\*) will be removed as soon as possible. Items with (\*\*) will issue a FATAL error (for the Nastran-to-Marc translator internal to Nastran) and Marc will not be “spawned” from Nastran unless NOERROR is entered on the SOL 600 statement.

- External superelements are supported. Other types of superelements are not currently supported.
- Scalar points are not supported.
- PBCOMP is not supported.
- CCONEAX is not supported.
- CBARAO is not supported.
- Output set definitions that contain grid or element numbers greater than the largest grid or element in the model will produce errors in Marc.
- Output set definitions may include the word BY as in output plot set definitions for use by Marc only. Nastran must be stopped using STOP=1 or one of the OUTR options since BY is a FATAL ERROR to Nastran.
- Nastran's CREEP entry must be changed to the MATVP entry.
- For orthotropic materials using MATORT, all shear moduli must be entered.
- SPOINTs are mapped to GRIDs with x, y, z coordinates at (0., 0., 0.)
- SLOAD and other scalar features are not supported.
- CELAS3 and CELAS4 are not supported.
- CLOAD is not supported.
- Fracture Mechanics is available.
- Aerodynamics is not supported.

- Bulk data entries with + or \* in column 73 must have an actual continuation card for most entries. Nastran does not require this, but the internal Marc translator does. (\*) .
- Slideline contact is not supported (BLSEG, BWIDTH, BFRIC, BCOMP, and BOUTPUT, if entered will cause FATAL ERRORS in Nastran).
- p-elements are not supported.
- Offsets are available for CBAR, CBEAM, CQUAD4 and CTRIA3 in all types of structural analysis (linear or nonlinear). The offsets must be specified in the global coordinate system (displacement output coordinate system) unless PARAM,MAROFSET is 1 (2 or 3, see description in [Parameter Descriptions, 784](#)). Offsets are available for CQUAD8 and CTRIA6 but only if all 8 or 6 grids are defined for these elements, respectively. If PARAM,MAROFSET,1 (2 or3) is included in the bulk data, the offsets will be incorporated using a new Marc feature that does not need extra grids or elements.
- For SOL 600, it is required that a Case Control LOAD or DLOAD entry be made for each subcase. If there are no subcases, one LOAD or DLOAD entry must be made.
- For SOL 600, it is required that all enforced displacements (other than motion of rigid contact surfaces applied using fields on the BCBODY entry) be applied using SPCD rather than SPC. The ID's of the SPCD must correlate with the Case Control LOAD or DLOAD entries.
- MPCs must be the same for all load cases.
- MAT10 is not presently supported.
- The following Solution Sequences are not presently supported: 107, 108, 110, 111, 114-116, 118, 144-146, 190, 200, 400 and 700 and will cause Severe Warnings (FATAL ERRORS) in the internal translator.
- DOMAINSOLVER is not supported. If this Executive Control statement is entered, and Nastran Implicit Nonlinear is requested by the SOL 600, ID statement, the DOMAINSOLVER request will be commented out internally by Nastran.
- IDs Grids, elements, properties, materials, etc. are limited to 9,999,999 unless one of the MRENUxx items is specified in which case the limit is 10 digits.
- CGAP does not completely map to Marc's gap element. The user should change all Nastran gaps to contact before running SOL 600. If the gaps are not changed to contact, some options will fail to translate as indicated by warning messages. Certain simple gaps translate as expected and will produce nearly the same results as standard Nastran solution sequences, but the user is responsible for ensuring that any model with gaps gives the behavior he expects when using SOL 600.
- Nastran MATS1 Mohr-Coulomb is mapped to Marc's Linear Mohr Coulomb option.
- Nastran MATS1 Drucker-Prager is mapped to Marc's Parabolic Mohr Coulomb option.
- Solid element composite output is not presently available using the OUTR options, It must be postprocessed directly using the t16 file - Patran is recommended.
- If layered output for Composite Structures is desired, the following bulk data parameters or bulk data entry, MARCOUT with LAYCODE of 1 or 2 should be included in the Bulk Data. If output for all layers is desired, set LAYCODE to zero and enter the following parameters:
  - param,mroutlay,N
  - param,marcslht,N

where N is the maximum number of layers in any composite PCOMP description. The preferred option is MARCOUT with LAYCODE=1.

- The rotational acceleration portion of RFORCE (RACC) is not supported and if entered will generate a Severe Warning and Marc will not be spawned.
- The following Case Control option for displacement/velocity/acceleration/spcforce/ mpcforce are not supported and will be ignored if entered:
  - SORT2, REAL, IMAG, PHASE, PSDF, ATOC, CRMS, RALL, RPRINT, RPUNCH, NORPRINT, CID, TM, RM
- Elements with mid-side nodes must have all mid-side nodes. For example CTETRA must either have 4 or 10 nodes.
- For PC systems, if SOL 600 is run from a command prompt (DOS box), if any old DOS programs are used prior to running SOL 600 the path where the job is being run is usually adjusted such that any names longer than 8 characters will be shortened (for example brake-squeal becomes BRAKE-~3). The continue options including brake squeal jobs will not work when this happens. Open a new command prompt and run SOL 600 before any old DOS programs are run in that window.
- The CID field on the RFORCE entry is not completely supported. If entered with a positive integer, the job will abort with a Severe Warning unless PARAM,MARCRCID is entered. PARAM,MARCRCID,1 may be used to ignore this field in which case R1,R2,R3 define the direction cosines of the rotation vector (see Marc Volume C, ROTATION A description) and the magnitude is given by  $A * \sqrt{R1^{**2} + R2^{**2} + R3^{**2}}$  [see RFORCE description for definitions of CID, A, R1, R2, R3 as well as Remark 16].
- Filenames entered on SOL 600 Bulk Data entries must be entered in small field fixed format, must be left-justified in the first applicable field and must be entered in lower case unless otherwise noted.
- For any jobs using the CONTINUE option or brake squeal, the jid must be entirely in lower case for Linux systems.
- Ixy of PBAR/PBEAM is ignored, if entered, for SOL 600.
- PARAM,MRDISCMB=1 must be used for models with multiple subcases with the same pressure loadings in each subcase. The program will automatically attempt to reset the default (mrdiscmb=0 to mrdiscmb=1 in such circumstances, however, it is recommended that the user does this himself).
- Some Bulk Data input entries are not checked as completely for SOL 600 as they are for other solution sequences, particularly those that apply only to SOL 600. In addition, certain checks for all types of entries, even those that can be used in other solution sequences may be made after Marc is spawned or the error messages not output until after Marc is spawned. For those cases, the error message, if any, will not be visible until Marc has finished and may not even be output unless one or more of the OUTR options is selected. Users should take special care that the SOL 600 input is free from errors and that no duplicates occur prior to running SOL 600. One way to do this which is highly recommended is to run a preliminary SOL 101 job with as much of the same input file as possible.
- For SOL 600, it is required that all enforced displacements (other than motion of rigid contact surfaces described by fields in BCBODY entries) be applied using SPCD rather than SPC. The ID's of the SPCD must correlate with the Case Control LOAD or DLOAD entries.

- If an .op2, .xdb, .f06, and/or punch file is requested using the OUTR option, static analyses must have “times” ranging from 0.0-1.0 for the first subcase, 1.0-2.0 for the second subcase, etc. If NLAUTO is used to change these times, the job will fail with an appropriate message. This means that if NLAUTO is used TFINAL (field 4) must always be 1.0.
- Concentrated masses are not considered in gravity loading for linear or nonlinear static analyses.
- Non-structural mass is ignored by SOL 600.

## SOL 700, ID

Executes MSC Nastran Explicit Nonlinear (SOL 700)

Format:

SOL 700, ID PATH= STOP= NP(or DMP700)= FSIDMP= INTELMPI=

Examples:

SOL 700,129 PATH=3 NP=4

(700,129 request nonlinear transient dynamics, path=3 requests use of the SOL 700 script called out in file sol700.pth, np=4 requests that 4 processors be used)

Summary:

SOL 700 is an Executive Control statement like SOL that activates an explicit nonlinear transient analysis integration scheme. The calculations will not be performed directly within MSC Nastran. Instead, SOL 700 will use a separate solver spawned from MSC Nastran. This client-server approach is similar to SOL 600, using Marc.

The SOL 700 statement will spawn an executable which is a 3D, explicit nonlinear analyses code DMP (distributed memory parallel processing domain decomposition) capabilities.

For ID=129 or NLTRAN, SOL 700 will generate an intermediate input data file, jid.dytr.dat, where “jid” is the name of the MSC Nastran input file without the extension). For example, if the MSC Nastran input file is named abcd.dat, (or abcd.bdf) then “jid”=abcd).

Unless specified differently using the STOP=3 option, the executable will be executed from MSC Nastran on any computer system capable of doing so (which includes most Linux systems and Windows systems). For it to run, it must be installed, properly licensed, and accessible from the directory where the MSC Nastran input data resides, MSC\_BASE must be provided in the environment.

Nastran SOL 700 Update:

Starting in MSC Nastran 2019.0 there is a change in the execution of MSC Nastran SOL 700. SOL 700 with LS-Dyna is no longer supported by MSC Nastran. Instead, a new explicit solver is introduced in MSC Nastran 2019.0. This version can be activated by adding VERSION = PRIMARY to SOL 700 in the input file or simply by leaving it blank as this will be the default. A new license feature will be needed to run this new version: NA\_Explicit\_Dytran.

The new SOL 700 will support both DMP and SMP parallelization. For this, the solver will pull licenses from the following license feature: NA\_Parallel.

Customers that have a lease agreement will no longer be able to run MSC Nastran SOL 700 jobs from version of MSC Nastran before 2018.0. The binaries needed were in the prior version located in msc20200/dytran directory of the MSC Nastran installation. However, this directory will be missing after the installation of MSC Nastran 2019.0.

Paid-up customers of MSC Nastran before 2018.0 can continue to use MSC Nastran SOL 700 jobs from version of MSC Nastran before 2018.0. For those customers, there will be no change in the FlexLM License features.



## Executive Control Parameters:

The required ID may be one of several valid solution sequence integers or names shown in [Table 2](#) for the SOL statement. Examples are 129 and NLTRAN.

The following solutions are available: 101, 106, 109, 129 (and their equivalent names).

All items on the SOL 700, ID after ID itself may be specified by environmental variables. This may be done any way environmental variables can be set. They may be set by the MSC Nastran user at run time or by the system administrator when MSC Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the [MSC Nastran 2020 Installation and Operations Guide](#). A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user's rc file, a local rc file or in a script used to submit MSC Nastran.

The following describes the various options for PATH. We suggest that PATH=3 for all computer systems.

### PATH=1 (Windows Only)

If PATH=1 is specified, MSC Nastran will determine the proper command to execute a **serial run**. To aid MSC Nastran in determining where Dytran is located, the **dynrun.pth** file must be located in the same directory where the MSC Nastran input file resides. The dynrun.pth file must contain one line providing the location (complete path) of the SOL 700 run script. A typical example of the line in the file dynrun.pth follows.

Windows                    c:\sol700\

A string is appended to this path to form the complete command used to execute the SOL 700 executable. "dytran jid=name.dytr.dat"

For Windows, MSC Nastran will spawn the external executable using the following command assuming the MSC Nastran input data is named enf2e.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

c:\sol700/dytran jid=enf2e.dytr.dat

### PATH=3 (All Systems)

If PATH=3 is specified, a script or batch file located in the same directory as the SOL 700 executable will be executed. The name of the executable is dytran (linux) or dytran.exe (Windows). This directory and name of the script is determined by the first line in a file named sol700.pth which must be in the same directory as the Nastran input file. Options are specified on subsequent lines of the sol700.pth file. For example, if Nastran is installed in C:\Program Files\MSC.Software\MSC\_Nastran\2020, the dytran.exe location is C:\Program Files\MSC.Software\MSC\_Nastran\2020\msc20200\dytran\win64\bin\dytran. To use sol700.pth file, the first line must be C:\Program Files\MSC.Software\MSC\_Nastran\2020\msc20200\dytran\win64\bin\dytran.

### Available PATH=3 options for Windows PC systems are as follows:

|          |   |
|----------|---|
| exe=     | The full path to the executable that is to be used.<br><br>Optional -- If exe= is omitted, the directory where the script or batch file resides (first line of sol700.pth) will be used and dytran for Linux and dytran.exe for windows will be appended. If exe= is used, it must be the second line in the sol700.pth file.   |
| nproc    | Number of processors. (only for DMP run)<br><br>(Default is to used NP on the SOL 700 line. If NP and nproc are omitted, the default is 1). For parallel execution, the directory where the MSC Nastran input file exists must be shared with read/write privileges. If wdir is used, it must also be shared (see below). The directory where the Dytran executable resides must also be shared for parallel execution. |
| ncpus    | Number of processors. (only for SMP)<br><br>(Default is not used.) If ncpus is greater than 1 in sol700.pth, Nastran SOL 700 automatically uses SMP capability.   |
| bat      | Run in background or foreground (Default).  |
| hlist    | Host file name. Name of a hostfile containing the same information as "machine"<br>The format of hostfile is as follows for the example for machine:<br>machine1 2<br>machine1 4  |
| intelmpi | To activate the Intel MPI set intelmpi equal to yes. Default is no. If intelmpi=yes is placed directly on the command line (not in sol700.pth) Intel MPI will be activated too.   |
| atb      | the name of atb file.   |
| imm      | the name of imm file.   |

A Windows example of the file sol700.pth for the PATH=3 case follows.

```
C:\MSC.Software\MSC_Nastran\2020\msc20200\dytran\win64\bin\dytran nastran
nproc=4
```

For the above example, MSC Nastran will create the following command to spawn the SOL 700 executable assuming your input file is named abcd.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

```
C:\Program Files\MSC.Software\MSC_Nastran\2020\msc20200\dytran\win64\bin\dytran
nastran nproc=6
```

### Available PATH=3 options for Linux systems follows:

|        |  |
|--------|--|
| debug  | Specifying debug=yes indicates if you want to keep scratch files and other debug information to investigate when a job fails to run. Default is no.  |
| exe    | The full path to the executable for Dytran that is to be used. (Optional)  |
| fsidmp | Specifying fsidmp=yes indicates to run the FSI Distributed Memory Parallel version. Default is no. If FSIDMP=YES is placed directly on the command line (not in sol700.pth) FSI Distributed Memory Parallel will be activated too. |

|       |  |
|-------|--|
| hlist | The (local) filename containing the hosts list. If this file is not given or not found, a default local hosts list is used. Note that the MPI universe in which the selected nodes and CPUs reside is expected to exist and be accessible (i.e., be booted). |
| nproc | Number of processors. (only for DMP run)<br><br>(Default is to use NP on the SOL 700 line. If NP and nproc are omitted, the default is 1.)<br>NOTE: The number of requested processes must be a power of 2.  |
| ncpus | Number of processors. (only for SMP)<br><br>(Default is not used.) If ncpus is greater than 1 in sol700.pth, Nastran SOL 700 automatically uses SMP capability.  |
| atb   | the name of atb file.  |
| imm   | the name of imm file.  |

A Linux example of the file sol700.pth for the PATH=3 case is as follows:

```
/app/msc/msc20200/dytran/linux64/bin/dytran nastran nproc=4
```

For the above example, MSC Nastran will create the a command similar to the following to spawn the SOL 700 executable assuming your input file is named abcd.dat

```
/app/msc/msc20200/dytran/linux64/bin/dytran nastran nproc=4
```

If PATH is not specified, the default search path will be used to locate the dytran executable. This version will be located in a subdirectory named dytran/machine below the MSC Nastran base directory (MSC\_BASE). Not all PATH=3 options are available using this default path option.

## STOP

STOP is an optional item. STOP is used to prevent execution of Dytran or prevent execution of MSC Nastran after IFP if so desired. The various options are as follows:

### STOP=1

If STOP=1 MSC Nastran will gracefully stop after IFP. This option is used to prevent MSC Nastran from performing its own solution (normally used when the solution is performed with ID=129).

### STOP=3

STOP=3 MSC Nastran is stopped after IFP and Dytran is not executed. This would be the normal STOP option if the user wants to examine the intermediate input file, make some changes and then execute Dytran manually.

The following files are potentially affected by the COPYR option:

**NP(or DMP700)=the Number of Processors**

NP(or DMP700)=the number of processors (domains) for parallel processing. The default is one. In order to use more than one domain, MPI, Lam, POE, or whatever parallel program is needed must be properly installed on all computers involved and a hostfile designating which computers are to be used for each domain

must have been setup prior to running the job. If NP>1, PATH=3 is used and a file named sol700.pth is located in the same directory as the MSC Nastran input data. The sol700.pth file should contain all commands necessary to run in parallel. This file must have execute permissions.

| Item       | Case Control Commands Available in SOL 700 |
|------------|--|
| \$         | Y  |
| BCONTACT   | Y  |
| BEGIN BULK | Y (Other BEGIN forms are not allowed)      |
| DLOAD      | Y  |
| ECHO       | Y  |
| ENDTIME    | Y  |
| ENDSTEP    | Y  |
| IC         | Y  |
| LOADSET    | Y  |
| \$S700     | Y  |
| SET        | Y  |
| SPC        | Y  |
| TITLE      | Y  |
| TSTEP      | Y (Same as )                               |
| TSTEPNL    | Y  |

The following summarizes the Bulk Data entries for SOL 700:

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| ABINFL  | Y                                      |             |
| ATBACC  | Y                                      |             |
| ATBJNT  | Y                                      |             |
| ATBSEG  | Y                                      |             |
| BARRIER | Y                                      |             |
| BCBODY  | Y                                      |             |
| BCBODY1 | Y                                      |             |
| BCBOX   | Y                                      |             |
| BCELIPS | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| BCGRID  | Y                                      |             |
| BCMATL  | Y                                      |             |
| BCONECT | Y                                      |             |
| BCONPRG | Y                                      |             |
| BCONPRP | Y                                      |             |
| BCPROP  | Y                                      |             |
| BCSEG   | Y                                      |             |
| BCTABL1 | Y                                      |             |
| BCTABLE | Y                                      |             |
| BIAS    | Y                                      |             |
| BJOIN   | Y                                      |             |
| BSURF   | Y                                      |             |
| CBAR    | Y                                      |             |
| CBEAM   | Y                                      |             |
| CDAMP1  | Y                                      |             |
| CDAMP1D | Y                                      |             |
| CDAMP2  | Y                                      |             |
| CDAMP2D | Y                                      |             |
| CELAS1  | Y                                      |             |
| CELAS1D | Y                                      |             |
| CELAS2  | Y                                      |             |
| CELAS2D | Y                                      |             |
| CHEXA   | Y(8 Nodes only)                        |             |
| CMARKB2 | Y                                      |             |
| CMARKN1 | Y                                      |             |
| COHFRIC | Y                                      |             |
| COMPUDS | Y                                      |             |
| CORD1C  | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| CORD1R  | Y                                      |             |
| CORD1S  | Y                                      |             |
| CORD2C  | Y                                      |             |
| CORD2R  | Y                                      |             |
| CORD2S  | Y                                      |             |
| CORD3R  | Y                                      |             |
| COUCOHF | Y                                      |             |
| COUOPT  | Y                                      |             |
| COUP1FL | Y                                      |             |
| COUPINT | Y                                      |             |
| COUPLE  | Y                                      |             |
| CPENTA  | Y(5 Nodes only)                        |             |
| CQUAD4  | Y                                      |             |
| CROD    | Y                                      |             |
| CSPR    | Y                                      |             |
| CTETRA  | Y (4 Audio Nodes only)                 |             |
| CTRIA3  | Y                                      |             |
| CTUBE   | Y                                      |             |
| CVISC   | Y                                      |             |
| CYLINDR | Y                                      |             |
| DAREA   | Y                                      |             |
| DETSPH  | Y                                      |             |
| DLOAD   | Y                                      |             |
| DYFSISW | Y                                      |             |
| DYPARAM | Y                                      |             |
| ENDDATA | Y                                      |             |
| ENDDYNA | Y                                      |             |
| EOSDEF  | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| EOSGAM  | Y                                      |             |
| EOSIG   | Y                                      |             |
| EOSJWL  | Y                                      |             |
| EOSMG   | Y                                      |             |
| EOSNA   | Y                                      |             |
| EOSPOL  | Y                                      |             |
| EOSTAIT | Y                                      |             |
| EOSUDS  | Y                                      |             |
| EULFOR  | Y                                      |             |
| EULFOR1 | Y                                      |             |
| EULFREG | Y                                      |             |
| FAILJC  | Y                                      |             |
| FAILMPS | Y                                      |             |
| FAILUDS | Y                                      |             |
| FFCONTR | Y                                      |             |
| FLOW    | Y                                      |             |
| FLOWC   | Y                                      |             |
| FLOWDEF | Y                                      |             |
| FLOWT   | Y                                      |             |
| FLOWUDS | N                                      |             |
| FORCE   | Y                                      |             |
| FORCE2  | Y                                      |             |
| FORCUDS | Y                                      |             |
| GBAG    | Y                                      |             |
| GBAGCOU | Y                                      |             |
| GRAV    | Y                                      |             |
| GRDSET  | Y                                      |             |
| GRID    | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| HEATLOS | Y                                      |             |
| HGSUPPR | Y                                      |             |
| HTRCONV | Y                                      |             |
| HTRRAD  | Y                                      |             |
| HYDSTAT | Y                                      |             |
| INCLUDE | Y                                      |             |
| INFLCG  | Y                                      |             |
| INFLFRC | Y                                      |             |
| INFLGAS | Y                                      |             |
| INFLHB  | Y                                      |             |
| INFLTNK | Y                                      |             |
| INFLTR  | Y                                      |             |
| INITGAS | Y                                      |             |
| LOAD    | Y                                      |             |
| LEAKAGE | Y                                      |             |
| LSEQ    | Y                                      |             |
| MAT1    | Y                                      |             |
| MAT2    | Y                                      |             |
| MAT8    | Y                                      |             |
| MATBV   | Y                                      |             |
| MATDEUL | Y                                      |             |
| MATEP   | Y                                      |             |
| MATF    | Y                                      |             |
| MATFAB  | Y                                      |             |
| MATHE   | Y                                      |             |
| MATORT  | Y                                      |             |
| MATRIG  | Y                                      |             |
| MATVE   | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| MESH    | Y                                      |             |
| MOMENT  | Y                                      |             |
| MOMENT2 | Y                                      |             |
| NLOUTUD | Y                                      |             |
| PBAR    | Y                                      |             |
| PBARL   | Y                                      |             |
| PBEAM   | Y                                      |             |
| PBEAML  | Y                                      |             |
| PBELT   | Y                                      |             |
| PCOMP   | Y                                      |             |
| PDAMP   | Y                                      |             |
| PELAS   | Y                                      |             |
| PELAS1  | Y                                      |             |
| PERMEAB | Y                                      |             |
| PERMGBG | Y                                      |             |
| PEULER  | Y                                      |             |
| PEULER1 | Y                                      |             |
| PLOAD   | Y                                      |             |
| PLOAD2  | Y                                      |             |
| PLOAD4  | Y (Continuation supported)             |             |
| PMARKER | Y                                      |             |
| PMINC   | Y                                      |             |
| PORFCPL | Y                                      |             |
| PORFGBG | Y                                      |             |
| PORFLOW | Y                                      |             |
| PORFLWT | Y                                      |             |
| PORHOLE | Y                                      |             |
| PORHYDS | Y                                      |             |

| Item    | Bulk Data Entries Available in SOL 700         | Fatal Error |
|---------|--|-------------|
| PORUDS  | Y  |             |
| PROD    | Y  |             |
| PSHELL  | Y  |             |
| PSHELL1 | Y  |             |
| PSOLID  | Y  |             |
| PVISC   | Y  |             |
| PVISC1  | Y  |             |
| RBE2    | Y  |             |
| RBJOINT | Y  |             |
| RELEX   | Y  |             |
| RFORCE  | (CID, METHOD, continuation line not supported) |             |
| SHREL   | Y  |             |
| SHRPOL  | Y  |             |
| SHRUDS  | Y  |             |
| SPC     | Y  |             |
| SPC1    | Y  |             |
| SPCADD  | Y  |             |
| SPHERE  | Y  |             |
| SURFINI | Y  |             |
| TABLED1 | Y  |             |
| TABLUDS | Y  |             |
| TIC     | Y  |             |
| TIC3    | Y (New Dytran type entry)                      |             |
| TICEL   | Y  |             |
| TICEUDS | Y  |             |
| TICEUL1 | Y  |             |
| TICREG  | Y  |             |
| TICVAL  | Y  |             |

| Item    | Bulk Data Entries Available in SOL 700 | Fatal Error |
|---------|--|-------------|
| TLOAD1  | Y                                      |             |
| TLOAD2  | N                                      |             |
| TODYNA  | Y                                      |             |
| TSTEP   | Y (Changed to TSTEPNL)                 |             |
| TSTEPNL | Y                                      |             |
| WALL    | Y                                      |             |
| WALL    | Y (New rigid wall entry)               |             |
| YLDHY   | Y                                      |             |
| YLDJC   | Y                                      |             |
| YLDMSS  | Y                                      |             |
| YLDPOL  | Y                                      |             |
| YLDRPL  | Y                                      |             |
| YLDSG   | Y                                      |             |
| YLDTM   | Y                                      |             |
| YLDUDS  | Y                                      |             |
| YLDVVM  | Y                                      |             |
| YLDZA   | Y                                      |             |

**SPARSEESOLVER****Sparse Solver Options**

Specifies options used in sparse solution of equations operations.

**Format:**

```
SPARSEESOLVER {target} ([COMPMETH = {cmeth}][ORDMETH = {ometh}],  
[FACTMETH = {fmeth}]MDTRATIO =  $\left( \left[ \begin{array}{l} \text{[NO]CHART} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nsegs} \end{array} \right] \right)$ ,  
 $\left[ \begin{array}{l} \text{[NO]TABLE} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nratios} \\ \text{NMAXRAT} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{maxratio} \\ \text{MAXRAT} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right]$ ,  
 $\left[ \begin{array}{l} \text{GRID} \\ \text{VALUE} \\ \text{BOTH} \end{array} \right] \right),$   
MDTSTATS =  $\left( \left[ \begin{array}{l} \text{[NO]CHART} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nsegs} \\ \text{nseg} \end{array} \right] \right)$ ,  
 $\left[ \begin{array}{l} \text{[NO]TABLE} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nmax} \\ \text{NMAXVAL} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{nmin} \\ \text{NMINVAL} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right]$ ,  
 $\left[ \begin{array}{l} \text{vmax} \\ \text{MAXVAL} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right] \left[ \begin{array}{l} \text{vmin} \\ \text{MINVAL} \\ \left\{ \begin{array}{l} T \\ R \end{array} \right\} \end{array} \right]$ ,  
 $\left[ \begin{array}{l} \text{GRID} \\ \text{VALUE} \\ \text{BOTH} \end{array} \right] )$ 
```

**Examples:**

- For the READ module, specify METIS reordering, and for the FRRD1 module, specify the UMFLU factorization method:

```
SPARSEESOLVER READ (ORDMETH=METIS)
SPARSEESOLVER FRRD1 (FACTMETH=UMFLU)
```

- Request matrix diagonal term ratio output:

```
SPARSEESOLVER DCMP ( MDTRATIO )
```

- Request matrix diagonal term ratio output in chart format for translational DOFs, and limit the number of printed diagonal ratios to 50:

```
SPARSEESOLVER DCMP ( MDTRATIO=(CHART, TABLET, NMAXRATT=50) )
```

- Request Matrix Diagonal Term statistics output for:

- Chart format for translational and rotational DOF
- Table format for translational dof limited to largest 10 terms greater than 1.0E+08

- Table format for translational dof limited to smallest 20 terms smaller than 1.0
- Table format for rotational dof limited to smallest 30 terms smaller than 100.0

```
SPARSEESOLVER DCMP (MDTSTATS=(CHART, TABLET, NMAXVALT=10,
MAXVALT=1.0E+08, NMINVALT=20, MINVALT=1.0, TABLER, NMINVALR=30,
MINVALR=100.00))
```

| Descriptor | Meaning  |
|------------|--|
| target     | The target application for the options. The target application choices are: CEAD, DCMP, DECOMP, MDSTAT, NLSOLVE, READ, SOLVE, FRRD1, and TRD1.   |
| cmeth      | Compression method to be used (Default = GRID).<br>The choices for compression method are: GRID, SUPER, GRDSUPER and NONE.   |
| ometh      | Ordering method to be used (Default = Automatic). The choices for ordering method are: AMF, BEND, MD, METIS, MMD, PRDMD, PRDMETIS, PRDSMPMS, and NONE. The default automatically selects among METIS, MMD and BEND   |
| fmeth      | Factorization method to be used (MSCLDL).<br>The choices for factorization method are: MSCLDL, MSCLU, UMFLU, NUMFLU, PRDLDL, and PRDLU. The default factorization method for symmetric matrices is MSCLDL. For unsymmetric matrices, the default is MSCLU. |
| NO         | No output is to be produced for the keyword.   |
| T          | The keyword applies to translational DOF only.   |
| R          | The keyword applies to rotational DOF only.  |
| nsegs      | Number of bars to be used in chart (Default = 0).  |
| nratios    | Number of diagonal term ratios to be included in table (Default = 25).   |
| maxratio   | Diagonal term ratio above which ratios are included in the table (Default = 1.0E+5).   |
| GRID       | Table output is sorted on GRID ID and component code.  |
| VALUE      | Sort table output on term values.  |
| BOTH       | Table is output once sorted on GRID ID and once on term value.   |
| nmax       | Number of largest values to print (Default = 25).  |
| nmin       | Number of smallest values to print (Default = 25).   |
| vmax       | All terms larger than vmax are printed (Default = 1.0E+10).  |
| vmin       | All terms smaller than vmin are printed (Default = 1.0).   |

### Remarks:

1. All of the keywords for the target application must be enclosed in parentheses.

2. The following table correlates target applications with available factorization methods.

| Module | MSCLDL | MSCLU | UMFLU | NUMFLU | PRDLDL | PRDLU |
|--------|--------|-------|-------|--------|--------|-------|
| CEAD   | Yes    | Yes   | No    | Yes    | No     | Yes   |
| DCMP   | Yes    | Yes   | No    | No     | Yes    | No    |
| DECOMP | Yes    | Yes   | No    | No     | No     | No    |
| FRRD1  | Yes    | Yes   | Yes   | No     | No     | Yes   |
| MDSTAT | Yes    | Yes   | No    | No     | No     | No    |
| NLSOLV | Yes    | Yes   | No    | No     | Yes    | No    |
| READ   | Yes    | No    | No    | No     | No     | No    |
| SOLVE  | Yes    | Yes   | No    | No     | No     | No    |
| TRD1   | Yes    | Yes   | Yes   | No     | No     | No    |

3. More than one SPARSESSOLVER entry may be used, so that one may specify different options for different target modules.
4. System cell 206 can specify both a reordering method and a compression method by adding the respective values. For example, to specify GRDSUPER compression method with BEND reordering, SYSTEM(206) would be set to 68 (since  $64+4=68$ ). Or, to specify SUPER compression with METIS reordering, SYSTEM(206) would be set to 136 (since  $128+8=136$ ).
5. Note that SYSTEM(166), SYSTEM(206) and SYSTEM(209) have precedence over the SPARSESSOLVER command, whether set on the submittal line or in DMAP.
6. System cell 166 can also be used to turn on extra diagnostic output from the sparse factorization by setting SYSTEM(166)=2.
7. The choices for ordering method are: AMF, BEND, MD, METIS, MMD, PRDMMD, PRDMETIS, PRDSMPMS, and NONE. The default automatically selects among METIS, MMD and BEND.
8. The UMFLU factorization method ignores COMPMETH and ORDMETH since it contains its own compression/reordering method.
9. The matrix diagonal term ratio (MDTRATIO) and value (MDTSTATS) keywords and options are used only by the DCMP target application. They will be ignored by other target applications.
10. The MDTRATIO and MDTSTATS keywords apply to both translational and rotational degrees of freedom unless modified by the T or R describer. Separate output is produced for each of the translational and rotational degrees of freedom in the matrix.
11. The MDTRATIO and MDTSTATS CHART option default produces a chart of values contained in powers of ten bandwidths. A specification for the nsegs describer causes the bandwidth to be internally computed to produce nsegs bars. Bars are produced for the bandwidths only if they contain terms.
12. MDTRATIO and MDTSTATS keywords are processed in the order given. It is possible for a keyword to modify the effects of a previously processed keyword. For example, the combination CHART, CHARTR=5 produces different outputs compared to CHARTR=5, CHART.

13. The MDTRATIO NMAXRAT descriptor limits the content of the TABLE output to the specified highest number of diagonal ratios that exceed the value of the MAXRAT descriptor.
14. MDTSTATS generates output for matrix diagonal term values. For the TABLE option, it produces a table containing the NMINVAL=nmin smallest terms smaller than MINVAL=vmin as well as the NMAXVAL=nmax largest terms larger than MAXVAL=vmax.
15. The ORDMETH option must be one of PRDMD, PRDMETIS, or PRDSMPMS for PRDLDDL and PRDLU.
16. The COMPMETH option is ignored for PRDLDDL and PRDLU.
17. The DOMAINsolver (RUNOPT=MULTIMST) option is not supported with PRDLDDL or PRDLU.
18. For PRDLDDL and PRDLU, if there is not sufficient memory for an in-core solver, then an out-of-core solver will be attempted and performance will be negatively affected.
19. The following special rules apply for PRDLDDL for NLSOLV:
  - a. The maximum number of threads used is given by DMP\*SMP, and PRDLDDL will choose a number of threads, up to this maximum, most suitable to the problem size.
  - b. PRDLDDL is supported only when FNT or PFNT method is used for nonlinear iteration (KMETHOD of NLPARM or NLSTEP).
  - c. If the problem fails to converge with PRDLDDL, then add MDLPRM,LMT2MPC,1 to the bulk data section in order to convert Lagrange multipliers to MPCs.
20. The following special rules apply for PRDLDDL for DCMP:
  - a. DMP is not supported for PRDLDDL for DCMP as it only gets parallelism from SMP
  - b. PRDLDDL automatically provides parallelization over multiple RHSs with SMP.
  - c. If PRDLDDL encounters a zero pivot issue (SFM 11332), the user should rerun with bulk data entry MDLPRM,PRDMTYPE,-2 to have Intel MKL Pardiso regard the matrix as indefinite, and entry MDLPRM,PRDWMTCH,0 in order to turn off weighted matching. Problems involving linear contact (BCONTACT) are particularly liable to creating indefinite systems. Turning off weighted matching will speed time to solution.
21. The following special rules apply for PRDLU for CEAD:
  - a. PRDLU is only supported for the CLAN option within CEAD (refer to CMETHOD option).
22. The following special rules apply for PRDLU for FRRD1:
  - a. The Krylov solver is not supported with PRDLU for FRRD1.
  - b. DMP is supported for PRDLU for FRRD1.
  - c. MKL Pardiso is not supported in the uncoupled solution algorithm for FRRD1.
23. For CEAD, the default solver for unsymmetric matrices is UMFPACK while for Hermitian matrices the default is MSCLU.
24. User specified factorization method (fmeth) via SPARSESSOLVER entry or SYSTEM(209) may be overwritten automatically due to matrix type or symmetry by MSC Nastran.

25. Several options for Intel MKL Pardiso are controlled using the [MDLPRM](#) bulk data entry. See the PRD\* parameters detailed in the MDLPRM section.

**TIME**

Sets Maximum CPU and I/O Time

Sets the maximum CPU and I/O time.

**Format:**

TIME[=]t1[,t2]

**Examples:**

1. The following example designates a runtime of 8 hours:

TIME 480

2. The following example designates 90 seconds:

TIME 1.5

| Descriptor | Meaning  |
|------------|--|
| t1         | Maximum allowable execution time in CPU minutes (Real or Integer>0; Default=1.89E9 seconds).                 |
| t2         | Maximum allowable I/O limit in minutes (Real or Integer>0; Default is infinity, which is machine dependent). |

**Remarks:**

1. The TIME statement is optional.
2. If t2 is specified then t1 must be specified.

190 |

## TIME

Sets Maximum CPU and I/O Time

# 5

# Case Control Commands

- Key to Descriptions
- The Case Control Section
- Case Control Command Summary
- Case Control Commands
- Case Control Applicability Tables
- OUTPUT(PLOT) Commands
- X-Y PLOT Commands
- OUTPUT(POST) Commands

## Key to Descriptions

| Brackets [ ] indicate that a choice of descriptors is optional. |   | Applied Load Output Request   | A brief description of the command is given.                                   |
|---|---|---|--|
| Requests the form and type of applied load vector output.       |   |   | Descriptors in uppercase letters are keywords that must be specified as shown. |
| <b>Format:</b>  |   | OLOAD [ ( SORT1 ) , PUNCH , [ or PHASE ] ] = { ALL n NONE }   | Descriptors in lower case letters are variables.                               |
| <b>Examples:</b>  |   | OLOAD = ALL<br>OLOAD(SORT1,PHASE)   | Braces { } indicate that a choice of descriptors is mandatory.                 |
|   |   | The default descriptors are shaded.   |  |
| Descriptors   | Meaning   |   |  |
| SORT1   | Output will be presented as a tabular listing of grid points for each load, frequency, or time, depending on the solution sequence.   | Parentheses are required if items inside the parentheses are specified.   | be presented as a tabular listing of frequency or time for each grid           |
| PRINT   | The printer will be the output medium.  |   |  |
| PUNCH   | Each of the descriptors is discussed briefly. Further details may be discussed under Remarks.   |   |  |
| REAL or IMAG  | Requests real or imaginary numbers. The use of either REAL or IMAG yields the same output.  |   |  |
| PHASE   | Requests polar coordinates. The angle is in degrees.  |   |  |
| ALL   | Applied loads for all nodes.  |   |  |
| NONE  | Applied load for none of the nodes.   |   |  |
| n   | Set identification of a previously appearing SET command. Only loads on points whose identification numbers appear on this SET command will be output. (Integer > 0).       |   |  |
| <b>Remarks:</b>   |   | The remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command, the command's relationship to other commands, restrictions and recommendations on its use, and further details regarding the descriptors. |  |
| 1.  | Both PRINT and PUNCH  | 1. Both PRINT and PUNCH   |  |
| 2.  | See the <i>MSC Nastran User's Manual</i> for more information on SORT1 and SORT2 formats and their descriptions.  | 2. See the <i>MSC Nastran User's Manual</i> for more information on SORT1 and SORT2 formats and their descriptions.   |  |
| 3.  | In a statics problem, a request for zero load components (e.g., zero force and zero moment) is equivalent to a request for zero load (e.g., zero force and nonzero moment). | 3. In a statics problem, a request for zero load components (e.g., zero force and zero moment) is equivalent to a request for zero load (e.g., zero force and nonzero moment).  |  |

## The Case Control Section

The Case Control Section has several basic functions. Specifically, it:

- Selects loads and constraints.
- Requests printing, plotting, and/or punching of input and out data (plotter commands are described in [Plotting](#) in the *MSC Nastran Reference Guide*).
- Defines the subcase structure for the analysis.

[Table 5-3](#) through [Table 5-8](#) at the end of this section indicate the applicability of each command in all solution sequences.

## Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters if the abbreviation is unique relative to all other commands. If not, the full name of the command (or at least the first eight characters if the name exceeds eight characters) must be specified in order to avoid errors. Each command is described as follows:

### Description

A brief sentence about the function of the command is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the command line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
SET 1 = 5, 6, 7, 8, 9,  
10 THRU 55
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command; the command's relationship to other commands, restrictions and recommendations on the command's use; and further descriptions of the describers.

## Case Control Command Summary

This section contains a summary of all Case Control commands under the following headings:

### Subcase Definition

#### 1. Output Request Delimiters

|   |  |
|---|--|
| <a href="#">ENDCARDS</a>  | Reactivates processing of Bulk Data entries (MSGMESH).   |
| <a href="#">OUTPUT (Case)</a>   | Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS. |
| <a href="#">OUTPUT(PLOT)</a>  | Beginning of structure plotter output request.   |
| <a href="#">OUTPUT(POST)</a><br>or<br><a href="#">SETS DEFINITION</a> | Indicates beginning of grid point stress output requests and/or p-element data recovery set definition.            |
| <a href="#">OUTPUT(XYOUT)</a><br>or<br><a href="#">OUTPUT(XYPLOT)</a> | Indicates beginning of curve plotter output request.   |
| <a href="#">OUTPUT(CARDS)</a>   | Suppresses processing of Bulk Data entries (MSGMESH).  |

#### 2. Subcase Delimiters

|                                |  |
|--------------------------------|--|
| <a href="#">REPCASE (Case)</a> | Delimits and identifies a repeated output request subcase. |
| <a href="#">SUBCASE (Case)</a> | Delimits and identifies a subcase.                         |
| <a href="#">SUBCOM (Case)</a>  | Delimits and identifies a combination subcase.             |
| <a href="#">SYM (Case)</a>     | Delimits and identifies a symmetry subcase.                |
| <a href="#">SYMCOM (Case)</a>  | Delimits and identifies a symmetry combination subcase.    |

#### 3. Subcase Control

|                               |  |
|-------------------------------|--|
| <a href="#">MASTER (Case)</a> | Allows the redefinition of a MASTER subcase.   |
| <a href="#">MODES (Case)</a>  | Repeats a subcase.   |
| <a href="#">SUBSEQ (Case)</a> | Gives the coefficients for forming a linear combination of the previous subcases.    |
| <a href="#">SYMSEQ (Case)</a> | Gives the coefficients for combining the symmetry subcases into the total structure. |

## Data Selection

### 1. Static Load Selection

|                                 |   |
|---------------------------------|---|
| <a href="#">DEFORM (Case)</a>   | Selects the element deformation set.  |
| <a href="#">CLOAD (Case)</a>    | Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only. |
| <a href="#">LOAD (Case)</a>     | Selects an external static loading set.   |
| <a href="#">LOADNAME (Case)</a> | Provides a name to be associated with a loading condition.  |

### 2. Dynamic Load Selection

|                                  |   |
|----------------------------------|---|
| <a href="#">DLOAD (Case)</a>     | Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.                                |
| <a href="#">LOADSET (Case)</a>   | Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands. |
| <a href="#">NONLINEAR (Case)</a> | Selects nonlinear dynamic load set for transient problems.  |

### 3. Constraint Selection

|                                     |   |
|-------------------------------------|---|
| <a href="#">AXISYMMETRIC (Case)</a> | Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.  |
| <a href="#">AUTOSPC (Case)</a>      | Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.   |
| <a href="#">BC (Case)</a>           | Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.  |
| <a href="#">DSYM (Case)</a>         | Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.   |
| <a href="#">MPC (Case)</a>          | Selects a multipoint constraint set.  |
| <a href="#">SPC (Case)</a>          | Selects a single-point constraint set to be applied.  |
| <a href="#">STATSUB (Case)</a>      | Selects the static solution to use in forming the differential stiffness for buckling, normal modes, complex eigenvalue, frequency response, and transient response analysis. |
| <a href="#">SUPORT1 (Case)</a>      | Selects the fictitious support set (SUPORT1 entries only) to be applied to the model.   |

### 4. Thermal Field Selection

|                                      |   |
|--------------------------------------|---|
| <b>TEMPERATURE (Case)</b>            | Selects the temperature set to be used in either material property calculations, or thermal loading in heat transfer and structural analysis. |
| <b>TEMPERATURE (Case) (INITIAL)</b>  | Selects initial temperature distribution for temperature-dependent material properties and heat transfer problems.                            |
| <b>TEMPERATURE (Case) (LOAD)</b>     | Selects temperature set for static thermal load.  |
| <b>TEMPERATURE (Case) (MATERIAL)</b> | Selects temperature set for temperature-dependent material properties.  |
| <b>TSTRU (Case)</b>                  | Defines a temperature set ID for a structures run based on a heat transfer subcase.   |

## 5. Static Solution Conditions

|                       |                                      |
|-----------------------|--------------------------------------|
| <b>SMETHOD (Case)</b> | Selects iterative solver parameters. |
|-----------------------|--------------------------------------|

## 6. Dynamic Solution Conditions

|                          |   |
|--------------------------|---|
| <b>CMETHOD (Case)</b>    | Selects complex eigenvalue extraction parameters.   |
| <b>FREQUENCY (Case)</b>  | Selects the set of forcing frequencies to be solved in frequency response problems.   |
| <b>FRF (Case)</b>        | Specifies data for Frequency Response Function (FRF) generation or for the FRF Based Assembly (FBA) process.  |
| <b>IC (Case)</b>         | Selects the initial conditions for direct transient analysis (SOLs 109, 129, and 159).  |
| <b>METHOD (Case)</b>     | Selects the real eigenvalue extraction parameters.  |
| <b>MODESELECT (Case)</b> | Requests a set of computed mode shapes for inclusion in dynamic analysis.   |
| <b>NSM (Case)</b>        | Selects nonstructural mass set.   |
| <b>RANDOM (Case)</b>     | Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.  |
| <b>RESVEC (Case)</b>     | Specifies options for and the calculation of residual vectors.  |
| <b>RGYRO (Case)</b>      | Activates gyroscopic effects and selects RGRYO or UNBALNC entries.  |
| <b>SDAMPING (Case)</b>   | Requests modal damping as a function of natural frequency in modal solutions, or viscoelastic materials as a function of frequency in direct frequency response analysis. |

[SMETHOD \(Case\)](#) Selects iterative solver override options in frequency response analysis.

[TSTEP \(Case\)](#) Selects integration and output time steps for linear or nonlinear transient analysis.

## 7. Direct Input Matrix Selection

[A2GG \(Case\)](#) Selects direct input fluid-structure coupling matrix.

[B2GG \(Case\)](#) Selects direct input damping matrices.

[B2PP \(Case\)](#) Selects direct input damping matrices.

[K2GG \(Case\)](#) Selects direct input stiffness matrices.

[K2PP \(Case\)](#) Selects direct input stiffness matrices, which are not included in normal modes.

[K42GG \(Case\)](#) Selects direct input structural damping matrices.

[M2GG \(Case\)](#) Selects direct input mass matrices.

[M2PP \(Case\)](#) Selects direct input mass matrices, which are not included in normal modes.

[MFLUID \(Case\)](#) Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

[P2G \(Case\)](#) Selects direct input load matrices.

[TFL \(Case\)](#) Selects the transfer function set(s) to be added to the direct input matrices.

## 8. Nonlinear Analysis

[ENDSTEP \(Case\)](#) Specifies final analysis step for SOL 700.

[ENDTIME \(Case\)](#) Specifies final analysis time for SOL 700.

[NLBUCK \(Case\)](#) Perform a nonlinear buckling analysis in SOL 400.

[NLHARM \(Case\)](#) Selects the parameters used for nonlinear harmonic response analysis.

[NLOPRM \(Case\)](#) Controls MSC Nastran nonlinear output, debug printout and debug POST.

[NLPARM \(Case\)](#) Selects the parameters used for nonlinear static analysis.

[NLSTEP \(Case\)](#) Selects integration and output time steps for static and transient nonlinear analysis in SOL 400.

[STEP \(Case\)](#) Defines and identifies a nonlinear analysis for SOL 400.

**SUBSTEP (Case)** Delineates and identifies a nonlinear analysis SUBSTEP for COUPLED analysis in SOL 400.

**TSTEPNL (Case)** Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, 600 and SOLs 400 and 700.

## 9. Aerodynamic Analysis

**AECONFIG (Case)** Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.

**AESYMXY (Case)** Aerodynamic XY plane of symmetry flag.

**AESYMXZ (Case)** Aerodynamic XZ plane of symmetry flag.

**AEXUREF (Case)** Define the reference aerodynamic extra point (controller) vector

**CSSCHD (Case)** Aerodynamic control surface schedule.

**DIVERG (Case)** Selects the divergence parameters in a static aeroelastic divergence problem.

**FMETHOD (Case)** Selects the parameters to be used in the aerodynamic flutter analysis.

**GUST (Case)** Selects the field in an aerodynamic response problem.

**TRIM (Case)** Selects trim variable constraints in static aeroelastic response.

## 10. Design Sensitivity and Optimization (SOL 200)

**ANALYSIS (Case)** Specifies the type of analysis being performed for the current subcase.

**AUXCASE (Case)** Delimits Case Control commands for an auxiliary model in SOL 200.

**AUXMODEL (Case)** References an auxiliary model for generation of boundary shapes in shape optimization.

**DESGLB (Case)** Selects the design constraints to be applied at the global level in a design optimization task.

**DESMOD (Case)** Assigns the design model parameter used to locate the associated datablocks for merging of two or more SOL 200 using the MultiOpt toolkit application.

**DESOBJ (Case)** Selects the DRESP1 or DRESP2 entry to be used as the design objective.

**DESSUB (Case)** Selects the design constraints to be used in a design optimization task for the current subcase.

**DESVAR (Case)** Selects a set of DESVAR entries for the design set to be used.

**DRSPAN (Case)** Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcase.

|                                |   |
|--------------------------------|---|
| <a href="#">DSAPRT (Case)</a>  | Specifies design sensitivity output parameters.                 |
| <a href="#">MODTRAK (Case)</a> | Selects mode tracking options in design optimization (SOL 200). |

## 11. .p-Element and p-Adaptivity Analysis

|  |  |
|--|--|
| <a href="#">ADAPT (Case)</a>           | Specifies whether or not the subcase is to participate in the p-adaptivity process.  |
| <a href="#">ADAPT (Case)</a>           | Specifies p-adaptivity control parameters.   |
| <a href="#">DATAREC (Case)</a>         | Requests form and type of output for p-version elements.   |
| <a href="#">OUTRCV (Case)</a>          | Selects the output options for p-elements defined on an OUTRCV Bulk Data entry.  |
| <a href="#">SET (Case)</a>             | Defines a set of element identification numbers only for the SURFACE and VOLUME commands (grid point stress) or the OUTRCV Bulk Data entry (p-element data recovery). This form of the SET command must and can only be specified after the SETS DEFINITION or OUTPUT(POST) command delimiter. |
| <a href="#">SETS DEFINITION (Case)</a> | Delimits the various type of commands under grid point stress and/or p-version element set definitions. This command is synonymous with OUTPUT(POST).  |
| <a href="#">VUGRID (Case)</a>          | Requests output of view grid and view element entries used in p-version element data recovery.   |

## 12. Adaptive Meshing

|                               |   |
|-------------------------------|---|
| <a href="#">HADAPT (Case)</a> | Specifies Mesh adaptivity control parameters. |
|-------------------------------|---|

## 13. Fluid-Structure Analysis

|                                   |   |
|-----------------------------------|---|
| <a href="#">A2GG (Case)</a>       | Selects a direct input fluid-structure coupling matrix.                             |
| <a href="#">ACFPRESULT (Case)</a> | Requests output of acoustic field point mesh results.                               |
| <a href="#">ACPOWER (Case)</a>    | Request output of the power radiated from the wetted surface.                       |
| <a href="#">FLSFSEL (Case)</a>    | Fluid-structure parameter collector for frequency and fluid superelement selection. |
| <a href="#">FLSPOUT (Case)</a>    | Fluid-structure parameter collector for mode participation.                         |
| <a href="#">FLSTCNT (Case)</a>    | Fluid-structure parameter collector for symmetry and force request.                 |

|                                  |  |
|----------------------------------|--|
| <a href="#">INTENSITY (Case)</a> | Requests output of acoustic intensity on wetted surface. |
| <a href="#">TRIMGRP (Case)</a>   | Selection of Trim Component(s)                           |

## 14. Nastran/ADAMS Interface

|                                  |  |
|----------------------------------|--|
| <a href="#">ADAMSMNF* (Case)</a> | Control for Nastran/ADAMS interface modal neutral file (MNF).<br><br>For MSC Nastran 2004, to ensure compatibility with the Adams msc2mnf took kit, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any output2 files generated will have a pre-MSC Nastran 2004 format. |
|----------------------------------|--|

## 15. Contact

|                                 |   |
|---------------------------------|---|
| <a href="#">BCHANGE (Case)</a>  | Selects the change of the definition contact bodies in contact analysis.                |
| <a href="#">BCMOVE (Case)</a>   | Contact body movement selection in contact analysis.                                    |
| <a href="#">BCONCHK (Case)</a>  | Requests contact model check.   |
| <a href="#">BCONTACT (Case)</a> | Requests contact analysis.  |
| <a href="#">BOUTPUT (Case)</a>  | Requests output for contact analysis.   |
| <a href="#">BSQUEAL (Case)</a>  | Selects data for brake squeal analysis  |
| <a href="#">UNGLUE (Case)</a>   | Selects the grids should use standard contact instead of glued contact in glued bodies. |

## 16. Monte-Carlo simulation

|                                |                                     |
|--------------------------------|-------------------------------------|
| <a href="#">MONCARL (Case)</a> | Control for Monte-Carlo simulation. |
|--------------------------------|-------------------------------------|

## Output Selection

### 1. Output Control

|                              |   |
|------------------------------|---|
| <a href="#">ECHO (Case)</a>  | Controls echo (i.e., printout) of the Bulk Data.  |
| <a href="#">ECHOOFF</a>      | Suppresses echo of Case Control.  |
| <a href="#">ECHOON</a>       | Reactivates echo of Case Control.   |
| <a href="#">LABEL (Case)</a> | Defines a character string that will appear on the third heading line of each page of printer output. |

|                                 |  |
|---------------------------------|--|
| <a href="#">LINE (Case)</a>     | Defines the maximum number of output lines per printed page.   |
| <a href="#">MAXLINES (Case)</a> | Sets the maximum number of output lines.   |
| <a href="#">PAGE (Case)</a>     | Causes a page eject in the echo of the Case Control Section.   |
| <a href="#">PLOTID (Case)</a>   | Defines a character string that will appear on the first frame of any plotter output.                                |
| <a href="#">POST (Case)</a>     | Activates postprocessor operations for selected output data.   |
| <a href="#">SKIP (Case)</a>     | Activates or deactivates the execution of subsequent commands in the Case Control Section (including plot commands). |
| <a href="#">SKIPON</a>          | Defines commands in the Case Control Section that are not to be processed.   |
| <a href="#">SKIPOFF</a>         | Resumes processing of commands in the Case Control Section.  |
| <a href="#">SUBTITLE (Case)</a> | Defines a subtitle that will appear on the second heading line of each page of printer output.                       |
| <a href="#">TITLE (Case)</a>    | Defines a character string that will appear on the first heading line of each page of MSC Nastran printer output.    |

## 2. Set Definition

|                                   |   |
|-----------------------------------|---|
| <a href="#">MAXMIN (Case)</a>     | Specifies options for max/min surveys of certain output data associated with grid points.   |
| <a href="#">OFREQUENCY (Case)</a> | Selects a set of frequencies for output requests.   |
| <a href="#">OMODES (Case)</a>     | Selects a set of modes for output requests.   |
| <a href="#">OTIME (Case)</a>      | Selects a set of times for output requests.   |
| <a href="#">PARTN (Case)</a>      | Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17). In SOLs 111 and 200, the PARTN Case Control command specifies the points at which modal participation factors are to be computed. |
| <a href="#">SET (Case)</a>        | Defines a set of element or grid point numbers to be plotted.   |
| <a href="#">SURFACE</a>           | Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.  |
| <a href="#">VOLUME</a>            | Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.   |

### 3. Physical Set Output Requests

|  |  |
|--|--|
| <a href="#">ACCELERATION (Case)</a>  | Requests the form and type of acceleration vector output.  |
| <a href="#">BOUTPUT (Case)</a>   | Requests line or 3D (SOLs 101, 106, 129, 153, 159, 400, and SOL 600) contact output.   |
| <a href="#">CMSENRGY (Case)</a>  | Requests the output of component (superelement) modal strain, kinetic, and damping energies.   |
| <a href="#">DISPLACEMENT (Case),<br/>VECTOR (Case), or<br/>PRESSURE (Case)</a> | Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.                                     |
| <a href="#">EDE (Case)</a>   | Requests the output of the energy loss per cycle in selected elements.   |
| <a href="#">EKE (Case)</a>   | Requests the output of the kinetic energy in selected elements.  |
| <a href="#">ELSDCON (Case)</a>   | Requests mesh stress discontinuities based on element stresses (see STRESS).   |
| <a href="#">ENTHALPY (Case)</a>  | Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).   |
| <a href="#">ESE (Case)</a>   | Requests the output of the strain energy in selected elements.   |
| <a href="#">EQUILIBRIUM (Case)</a>   | Requests equilibrium force balance output.   |
| <a href="#">FATIGUE (Case)</a>   | Request a fatigue analysis with life/damage output in SOLs 101, 103, 108, 111, or 112.   |
| <a href="#">FLUX (Case)</a>  | Requests the form and type of gradient and flux output in heat transfer analysis.  |
| <a href="#">FORCE (Case) or<br/>ELFORCE</a>                                    | Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command. |
| <a href="#">GPFORCE (Case)</a>   | Requests grid point force balance at selected grid points.   |
| <a href="#">GPKE (Case)</a>  | Requests the output of the kinetic energy at selected grid points in normal modes analysis only.   |
| <a href="#">GPSDCON (Case)</a>   | Requests mesh stress discontinuities based on grid point stresses (see GPSTRESS).  |
| <a href="#">GPSTRAIN (Case)</a>  | Requests grid points strains for printing only.  |
| <a href="#">GPSTRESS (Case)</a>  | Requests grid point stresses for printing only.  |
| <a href="#">GVECTOR (Case)</a>   | Requests the form and type of g-set eigenvector output in SOLs 200 and 400.  |
| <a href="#">HDOT (Case)</a>  | Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).   |

|   |  |
|---|--|
| <a href="#">HISTOGRAM (Case)</a>              | Requests resultant stress/strain time history and/or rainflow cycle histogram output from entities of a FATIGUE output request in SOL 101, 103, and 112. |
| <a href="#">MCFRACTION (Case)</a>             | Requests modal contribution fractions output.  |
| <a href="#">MODALKE (Case)</a>                | Requests modal kinetic energy output.  |
| <a href="#">MODALSE (Case)</a>                | Requests modal strain energy output.   |
| <a href="#">MEFFMASS (Case)</a>               | Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.                     |
| <a href="#">MPCFORCES (Case)</a>              | Requests the form and type of multipoint force of constraint vector output.  |
| <a href="#">NLSTRESS (Case)</a>               | Requests the form and type of nonlinear element stress output in SOL 106.  |
| <a href="#">NOUTPUT (Case)</a>                | Requests physical output in cyclic symmetry problems.  |
| <a href="#">OLOAD (Case)</a>                  | Requests the form and type of applied load vector output.  |
| <a href="#">RCROSS (Case)</a>                 | Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.                                      |
| <a href="#">SPCFORCES (Case)</a>              | Requests the form and type of single-point force of constraint vector output.  |
| <a href="#">STRAIN (Case)</a>                 | Requests the form and type of strain output.   |
| <a href="#">STRESS (Case) or<br/>ELSTRESS</a> | Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.  |
| <a href="#">STRFIELD (Case)</a>               | Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.  |
| <a href="#">THERMAL (Case)</a>                | Requests the form and type of temperature output.  |
| <a href="#">VELOCITY (Case)</a>               | Requests the form and type of velocity vector output.  |

#### 4. Solution Set Output Requests

|                                   |   |
|-----------------------------------|---|
| <a href="#">AEROF (Case)</a>      | Requests the aerodynamic loads on aerodynamic control points.   |
| <a href="#">APPRESSURE (Case)</a> | Requests the aerodynamic pressures in static aeroelastic response.  |
| <a href="#">HARMONICS (Case)</a>  | Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems. |
| <a href="#">HOUTPUT (Case)</a>    | Requests harmonic output in cyclic symmetry problems.   |
| <a href="#">MPRES (Case)</a>      | Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.  |

|                                      |  |
|--------------------------------------|--|
| <a href="#">NLLOAD (Case)</a>        | Requests the form and type of nonlinear load output for transient problems.  |
| <a href="#">SACCELERATION (Case)</a> | Requests the form and type of solution set acceleration output.  |
| <a href="#">SDISPLACEMENT (Case)</a> | Requests the form and type of solution set displacement output.  |
| <a href="#">SVECTOR (Case)</a>       | Requests the form and type of solution set eigenvector output.   |
| <a href="#">SVELOCITY (Case)</a>     | Requests the form and type of solution set velocity output.  |
| <a href="#">TRIMF (Case)</a>         | Specifies options for the output of trim loads from a static aeroelastic analysis as FORCE/MOMENT Bulk Data entries. |

## 5. Model Checkout

|                                    |   |
|------------------------------------|---|
| <a href="#">ELSUM (Case)</a>       | Requests a summary of element properties for output.  |
| <a href="#">GROUNDCHECK (Case)</a> | Requests grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.  |
| <a href="#">WEIGHTCHECK (Case)</a> | At each stage of the mass matrix reduction, computes rigid body mass and compares with the rigid body mass t the g-set. |

## Superelement Control

|                                   |   |
|-----------------------------------|---|
| <a href="#">EXTDRIN (Case)</a>    | Requests the job to perform an external superelement data recovery restart. Also specifies the storage media of the boundary solution data.   |
| <a href="#">EXTDROUT (Case)</a>   | Requests the job to store external superelement boundary displacements and column labels; e.g., eigenvalues, forcing frequencies, time steps. Also specifies the storage media of the boundary solution data. |
| <a href="#">EXTSEOUT (Case)</a>   | Specifies the data to be saved for an external superelement, and the medium on which the data is to be saved.   |
| <a href="#">SEALL (Case)</a>      | Specifies the superelement identification numbers of phase 1 processing wherein all matrices and loads are generated and assembled. Controls execution of the solution sequence.                              |
| <a href="#">SEDR (Case)</a>       | Specifies the superelement identification numbers for which data recovery will be performed.  |
| <a href="#">SEDV (Case)</a>       | Specifies the superelement identification numbers for which the design variables will be processed.   |
| <a href="#">SEEINCLUDE (Case)</a> | Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.  |

|                                    |   |
|------------------------------------|---|
| <a href="#">SEFINAL (Case)</a>     | Specifies the superelement identification number of the final superelement to be assembled.   |
| <a href="#">SEKREDUCE (Case)</a>   | Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.   |
| <a href="#">SELGENERATE (Case)</a> | Specifies the superelement identification numbers for which static loads will be generated.   |
| <a href="#">SELREDUCE (Case)</a>   | Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.   |
| <a href="#">SEMGENERATE (Case)</a> | Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.  |
| <a href="#">SEMREDUCE (Case)</a>   | Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices are assembled and reduced. |
| <a href="#">SERESP (Case)</a>      | Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.  |
| <a href="#">SOLUTION (Case)</a>    | Selects the solution ID for a 3 <sup>rd</sup> step external superelement data recovery restart in SOL 400.  |
| <a href="#">SUPER (Case)</a>       | Assigns a subcase(s) to a superelement or set of superelements.   |

## Miscellaneous

|                                   |   |
|-----------------------------------|---|
| \$                                | Used to insert comments into the input file. Comment statements may appear anywhere within the input file.          |
| <a href="#">BEGIN BULK (Case)</a> | Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.                         |
| <a href="#">INCLUDE (Case)</a>    | Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file. |
| <a href="#">NSM (Case)</a>        | Request nonstructural mass distribution selection.  |
| <a href="#">OUTPUT (Case)</a>     | Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.  |
| <a href="#">PARAM (Case)</a>      | Specifies values for parameters.  |
| <a href="#">POST (Case)</a>       | Controls selection of data to be output for postprocessing.   |
| <a href="#">RIGID (Case)</a>      | Selects type of rigid element formulations to be used.  |

## Case Control Commands

### \$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

For SOL 700 only, the string '\$S700' (starting in column 1 with a blank in column 6) may be used to transfer any Case Control string directly to Dytran. For example:

```
$S700 ELOUT (TEST1)=XVEL,YVEL,ZVEL,DENSITY,SIE,PRESSURE,fmat  
will become
```

```
ELOUT (TEST1)=XVEL,YVEL,ZVEL,DENSITY,SIE,PRESSURE,fmat
```

in the Case Control Section of the Dytran input (jid.dytr.dat) file. Please see descriptions of '\$S700' Case Control entries elsewhere in this section.

**Format:**

\$ followed by any characters out to column 80.

**Example:**

```
$ TEST FIXTURE-THIRD MODE
```

**Remarks:**

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

## \$S700

## Direct Case Control Command for SOL 700

All entries which start with \$S700 in the Case Control Section will be passed directly to the SOL 700 solver.

Import of Euler results and the control of output generation are using this entry:

### Output Control:

No need to use the Nastran classical case control output (STRESS,DISP,...). This option controls ARC (state file) and THS (time history file) output files.

You need to specify the following for a complete output specification:

1. Type of the file
2. What entities (e.g., grid points, elements, rigid bodies, etc.)
3. What results are output
4. How often it is saved
5. How often data is written

### Format:

```
$S700 TYPE(logical_file) = ARCHIVE
$S700 Entity Type(logical_file) = setid
$S700 SET setid = idi
$S700 Entity Var(logical_file) = var
$S700 TIMES(logical_file) = ti
$S700 STEPS(logical_file) = stepi
$S700 SAVE(logical_file) = n
```

### Example:

```
$S700 TYPE (elements) = ARCHIVE
$S700 ELEMENTS (elements) = 1
$S700 SET 1 = 16805 THRU 16875 16877 THRU 16947 16949 THRU 18080 ,
$S700      59 THRU 945 1067 1068 1069 1070 1181 THRU 1412 ,
$S700      1423 THRU 2965 3076 THRU 3930 4041 THRU 4895 ,
$S700      5006 THRU 5860 5971 THRU 7744 10855 THRU 12025 ,
$S700      12138 THRU 12990 13103 THRU 13955 14068 THRU 14920 ,
$S700      15033 THRU 15885 15998 THRU 16875 16877 THRU 16947 ,
$S700      16949 THRU 18080 16805 THRU 16875 16877 THRU 16947 ,
$S700      17005 THRU 18080 16949 THRU 17004 BY 1
$S700 ELOUT (elements) = EFFSTS, PRESSURE, FMAT, YMOM ,
$S700      QDIS,
$S700      MASST
$S700 TIMES (elements) = 0,THRU,END,BY,4e-4
$S700 SAVE (elements) = 1
```

| Descriptor   | Meaning  |
|--------------|--|
| logical_file | The logical name of the file to which the user output is written. The logical name may not contain any spaces or special characters and must be restricted to 8 characters or less. (Characters; Required)   |
| setid        | Number of a SET command. Only data for elements that appear in the set are output. (Integer > 0; Required)   |
| Idi          | Identification numbers at which the output is requested. "THRU" and "BY"; for the "THRU" and "BY" options, respectively. See Remark 2. (Integer > 0 or characters; Required)                                 |
| Var          | Variable name to be output. See Remark 5. (Characters; Required)   |
| Ti           | Times at which output is required. "THRU" and "BY"; for the "THRU" and "BY" options, respectively. "END" indicates the end time of analysis. See Remarks 3. and 4. (Real $\geq 0.0$ or characters; Required) |
| stepi        | Steps at which output is required. "THRU" and "BY"; for the "THRU" and "BY" options, respectively. See Remarks 3. and 4. (Integer $\geq 0$ or characters; Required)  |
| N            | The number of times an output file is written before it is closed and saved. (Integer > 0; Required)   |

### Remarks:

1. Default of ARC output control is set to the ARC file generation of all Eulerian elements with all available variables.
2. Continuation lines are supported for SET, ELOUT, TIMES and STEPS. The continuation lines must start with at least two spaces after the \$S700 string. A comma (,) at the end of a line signifies that the next line is a continuation.
3. Either STEPS or TIMES must be specified. STEPS and TIMES can not be used for the same logical\_file.
4. A list of times or steps should be in ascending order.
5. Check [Chapter 11: Outputting Results](#) in *MSC Nastran SOL700 User's Guide*. It includes all Entity types, Entity variables and Variable names.

### Import Euler results:

Specifies an Euler archive that was created during a previous simulation and is used as input for a transient analysis. The Euler archive is mapped onto a set of Euler elements that can equal in size or either finer or coarser.

### Format:

\$S700 EULINIT, filename, CYCLE, MESH-ID

### Example:

\$S700 EULINIT, AXISYM\_ALLEULER\_0.ARC, 60

| Descriptor | Meaning                                  |
|------------|--|
| filename   | The filename of the ARC file to be used. |
| CYCLE      | Cycle number.                            |
| MESH-ID    | The MESH ID of the target elements.      |

#### Remarks:

1. The target elements are the elements defined in the follow-up run and are the elements that will be initialized using the import archive.
2. Both defined and imported Euler elements need to be orthogonal in the global system.
3. MESH-ID enables support for multiple Euler domains. If MESH-ID is not set the import archive will be mapped onto all Euler elements.
4. For multi-material Euler analyses with multiple Eulerian materials all material variables in the import archive require the material number. These material variables are MASS, DENSITY, SIE, FMAT. The required list of variables for a MMHYDRO run are: MASSXX, SIEXX, FMATXX, XVEL, YVEL, ZVEL. Here XX denotes the material number. FOR MMSTREN the variables TXX, TYY, TZZ, TXY, TYZ, TZX, EFFPLS have to be added. If the multi-material run uses only one Eulerian material then the material numbers can be left out.
5. For multi-material Euler analyses with [EOSIG](#), the following has to be added for the IG materials: MASS-EXX, MASS-PXX, RHO-EXX, RHO-PXX, IGBURNXX, FMAT-PXX, FMAT-PXX, SIE-EXX, and SIE-PXX. Here XX denotes the material number
6. For the single-material Hydro Euler solver the required list is MASS, DENSITY, SIE, FMAT, and FVUNC.
7. IF FVUNC is not included in the Import archive it is assumed that all elements in this archive are fully uncovered. It is allowed to import such an archive in a simulation with a coupling surface. In this follow up simulation the target elements can have uncover fractions different from one. In this case conservative quantities of imported elements are reduced by the uncover fraction of the target element. This is to avoid unwanted pressure increases. It simply means that any mass of the import archive that is located in the covered part of the target elements is thrown away. As a result not all mass in the import archive is mapped to the target elements. How much of the mass of the import archive is mapped is shown in the out file.
8. In the OUT file, a summary is shown of all variables that are mapped.
9. In the follow-up run, the cycle and time are taken from the import archive. The results of the first cycle of the follow-up run are determined from remapping only and has not gone through an equation of state yet. This will happen in the next cycle
10. If needed the remapping can be checked by doing only one additional cycle in the follow-up run with a quite small time step. Then, the follow-up OUT file shows two cycles and the results should be almost identical to the results of the import archive. Also, material summaries in the OUT file between first run and follow-up run should be identical. The only exceptions are the summaries of momentum, kinetic energy, and total energy per material. For these three quantities, only the total amounts will remain constant between first and follow-up run.

11. To remap a spherical symmetric or an axial symmetric Euler archive, the DYPARAMS [DYPARAM,SPREMAP](#) and [DYPARAM,AXREMAP](#) have to be used. Also, the remapping of a spherical symmetric Euler domain onto a 2-D axial symmetric Euler domain is supported.

## A2GG

Selects a Direct Input Fluid-Structure Coupling Matrix

Selects a direct input fluid-structure coupling matrix.

**Format:**

A2GG = name

**Example:**

A2GG = AGG0

| Descriptor | Meaning  |
|------------|--|
| name       | Name of a fluid-structure coupling matrix that is input on the DMIG Bulk Data entry. |

**Remarks:**

1. DMIG entries will not be used unless selected by the A2GG Case Control command.
2. This entry must be above subcase level or in the first subcase.
3. If the A2GG Case Control command selects a DMIG entry, then Nastran will add the selected fluid-structure coupling matrix to the computed coupling matrix. To replace the computed coupling matrix with the selected A2GG matrix, set PARAM,ASCOUP,NO. The user may still define panels with the panel selection procedure.
4. When filling out the DMIG entries: IFO = 1, NCOL = g-size, GJ-column index corresponds to fluid points, CJ = 0, Gi-row index corresponds to structural points, Ci-corresponds to DOF, Ai-the area values.
5. A2GG is supported in dynamic solutions with fluid-structure coupling.
6. Only one A2GG command should be used. It must appear above any subcase structure or in the first subcase only.

**ACCELERATION****Acceleration Output Request**

Requests form and type of acceleration vector output.

**Format:**

$$\begin{aligned}
 ACCELERATION & \left[ \left( \begin{matrix} SORT1 \\ SORT2 \end{matrix} \right), \left( \begin{matrix} PRINT, PUNCH \\ PLOT \end{matrix} \right), \left( \begin{matrix} REAL \text{ or } IMAG \\ PHASE \end{matrix} \right), \left( \begin{matrix} PSDF, ATOC, CRMS \\ \text{or RALL} \end{matrix} \right), \right. \\
 & \quad \left. \left[ \begin{matrix} RPRINT \\ NORPRINT, RPUNCH \end{matrix} \right], CID \right] \\
 & = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

**Examples:**

ACCELERATION=5

ACCELERATION(SORT2, PHASE)=ALL

ACCELERATION(SORT1, PRINT, PUNCH, PHASE)=17

ACCELERATION(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20

ACCELERATION(PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG

Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

| Descriptor | Meaning   |
|------------|---|
| PHASE      | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| PSDF       | Requests the power spectral density function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.      |
| ATOC       | Requests the autocorrelation function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.             |
| CRMS       | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5. |
| RALL       | Requests all of PSDF, ATOC and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.               |
| RPRINT     | Writes random analysis results in the print file (Default).   |
| NORPRINT   | Disables the writing of random analysis results in the print file.  |
| RPUNCH     | Writes random analysis results in the punch file.   |
| CID        | Requests printing of output coordinate system ID in printed output file (.f06).   |
| ALL        | Accelerations at all points will be output.   |
| n          | Set identification of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output (Integer > 0).  |
| NONE       | Accelerations at no points will be output.  |

**Remarks:**

1. Acceleration output is only available for transient and frequency response problems. Acceleration is only available for transient and frequency response problems and when response spectra is requested in eigenvalue analysis.
2. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.
3. ACCELERATION=NONE allows overriding an overall output request.
4. Acceleration results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
5. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
6. Note that the CID keyword affects only grid point related output, such as displacement (DISP), velocity (VELO), acceleration (ACCEL), OLOAD, SPCforce (SPCF), and MPCforce (MPCF). In addition, the CID keyword needs to appear only once in a grid-related output request anywhere in the Case Control Section to turn on the printing algorithm.

**ACFPMRESULT**

## Acoustic Field Point Mesh Results Output Request

Requests output of field point mesh results. This Case Control command can be used in SOL 108 and SOL 111 only.

Format:

$$\begin{aligned} ACFPMRESULT & \left( \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, \begin{bmatrix} VELOCITY = \left\{ \begin{array}{l} YES \\ NO \end{array} \right\} \end{bmatrix} \right. \\ & \left. \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix}, \begin{bmatrix} POWER = \left\{ \begin{array}{l} YES \\ NO \end{array} \right\} \end{bmatrix} \right) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\} \end{aligned}$$

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as tabular listing of grid points for each excitation frequency (Default). |
| SORT2      | Output will be presented as a tabular listing of excitation frequencies for each grid point.        |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |   |
|--------------|---|
| VELOCITY     | Requests output of particle velocities (Default = NO).  |
| REAL or IMAG | Requests rectangular format (real or imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| ALL          | Radiated power will be processed for the wetted surface and all panels.   |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                            |
| POWER        | Requests output of power through field point mesh (Default = YES).  |
| ALL          | Results of all field point meshes, AFPMIDs, will be processed.  |

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a previously defined set of field point mesh identifiers, AFPMIDs, Results will be processed for the field point meshes in this set only. |
| NONE       | Field point mesh results will not be processed. ACFPMRESULT = NONE overrides an overall request.  |

**Remark:**

1. If the acoustic model references multiple PACINF entries that do not have coincident pole locations and if the acoustic field point meshes contain element data, then a PACINFID parameter entry is required in the bulk data for each mesh to identify the PACINF bulk data entry that is used to define the location of the pole. Data recovery operations require that the element surface normal vector point away from the pole location.
2. The member of set for ACFPMRESULT must be the IDs of 'BEGIN AFPM='; not GRID IDs or element IDs under a 'BEGIN AFPM='.

**ACPOWER****Acoustic Power Output Request**

Requests output of the power radiated from the wetted surface.  
This Case Control command can be used in SOL 108 and SOL 111 only.

Format:

$$ACPOWER \left[ \left[ \begin{matrix} SORT1 \\ SORT2 \end{matrix} \right], \left[ \begin{matrix} PRINT, PUNCH \\ PLOT \end{matrix} \right], [CSV = unit] \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of panels for each excitation frequency.            |                   |                      |
| SORT2            | Output will be presented as a tabular listing of excitation frequencies for each panel (Default). |                   |                      |
|                  | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| CSV  | Results will be written to a .CSV file. See Remark 1.   |
| unit | Unit of the .csv file as used on the ASSIGN statement.  |
| ALL  | Radiated power will be processed for the wetted surface and all panels.   |
| n    | Set identification of a previously defined set of panels. Radiated power will be processed for the wetted surface and all panels in the referenced set. |
| NONE | Radiated power will not be processed.<br>ACPOWER = NONE overrides an overall request.   |

#### Remarks:

- If output to an .CSV file is requested, the file must be assigned with logical key “USERFILE” and FORM=FORMATTED, e.g.,
 

```
ASSIGN USERFILE = 'myfile.csv' UNIT=50 FORM=FORMATTED STATUS=NEW
```

## ACTIVAT

### Elements to be Reactivated in SOL 600

Indicates which Bulk Data ACTIVAT entry is used to control the elements to be activated in this subcase. This entry may only be used in SOL 600.

Format:

ACTIVAT=N

Example:

ACTIVAT=3

| Descriptor | Meaning  |
|------------|--|
| N          | ID of a matching Bulk Data ACTIVAT entry specifying the elements to be reactivated for this subcase. |

Remarks:

1. Different sets of elements can be reactivated during different subcases using this Case Control command.
2. The elements specified in the matching ACTIVAT Bulk Data entry must currently be in a deactivated state.

**ACTRIM**

## ACTRAN Trimmed Material Matrices for SOL 108/111

Select ACTRAN trimmed material matrices.

**Format:**

ACTRIM = name1, name2, ... namen

**Example:**

ACTRIM = FLOOR\_F, FLOOR\_R, DASH

SET 10 = FR\_LH, RR\_LH

ACTRIM = 10

| Descriptor | Meaning   |
|------------|---|
| namei      | Name of the ACTRAN trimmed material matrices that is input on the ACTRIM bulk data entry, or name list. |

**Remarks:**

1. This entry must be above subcase level or in the first subcase.
2. If the ACTRIM Case Control command selects ACTRIM bulk data entries, Nastran will add the selected ACTRAN matrices to fluid-structure coupling problem in all subcases.
3. ACTRIM is supported in frequency response analysis for fluid-structure coupling problem and the frequency dependent algorithm will be adopted automatically.
4. PARAM, ACSYM, YES should be set for ACTRIM (default).
5. The effect of ACTRIM will be considered in standard frequency response analysis and participation factor analysis by PFMODE, PFPANEL and PFGRID.

**ADACT****p-Adaptivity Subcase Selection**

Specifies whether or not the subcase is to participate in the p-adaptivity process.

**Format:**

$$ADACT = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

ADACT=NONE

ADACT=10

| Descriptor | Meaning  |
|------------|--|
| ALL        | All subcases will participate in the error analysis.   |
| n          | The first n modes in a normal modes analysis will participate in the error analysis (Integer > 0). |
| NONE       | The current subcase will not participate in the error analysis.                                    |

**Remarks:**

1. ADACT is processed only when an adaptive analysis is requested. An ADAPT Case Control command must be present in order to have an p-adaptive analysis.
2. In a static analysis, ADACT=n is equivalent to ADACT=ALL, and ALL means that the results of all subcases will be included in the error analysis. When ADACT=NONE in any subcase, the results of that subcase are excluded from the error analysis and adaptivity.
3. In an eigenvalue analysis, ALL means that the results of all the modes will be included in the error analysis.
4. Only one ADACT command may be specified per subcase.

**ADAMSMNF\*****Control for Nastran/ADAMS Interface**

Control for Nastran/ADAMS Interface modal neutral file (.mnf)

Format:

$$\begin{aligned}
 ADAMSMNF & \left[ FLEXBODY = \begin{cases} NO \\ YES \end{cases} \right], \left[ FLEXONLY = \begin{cases} YES \\ NO \end{cases} \right], \\
 & \left[ ADMCHECK = \begin{cases} NO \\ YES \end{cases} \right], \left[ ADMOUT = \begin{cases} NO \\ YES \end{cases} \right], \\
 & \left[ OUTGSTRS = \begin{cases} YES \\ NO \end{cases} \right], \left[ OUTGSTRN = \begin{cases} YES \\ NO \end{cases} \right], \\
 & \left[ OUTSTRS = \begin{cases} NO \\ YES \end{cases} \right], \left[ OUTSTRN = \begin{cases} NO \\ YES \end{cases} \right] \\
 & \left[ V1ORTHO = \begin{cases} -1.0 \\ value1 \end{cases} \right], \left[ V2ORTHO = \begin{cases} 1.0e8 \\ value2 \end{cases} \right], \\
 & \left[ MINVAR = \begin{cases} PARTIAL \\ CONSTANT \\ FULL \\ NONE \\ RIGID \end{cases} \right], \left[ PSETID = \begin{cases} NONE \\ setid \\ plotel \\ ALL \end{cases} \right] \left[ EXPORT = \begin{cases} MNF \\ DB \\ BOTH \end{cases} \right] \left[ MONITOR = \begin{cases} YES \\ NO \end{cases} \right]
 \end{aligned}$$

Example(s):

ADAMSMNF FLEXBODY = YES

**Note:**

\*Nastran/ADAMS modal stress recovery (MSR) interface is also available. See Remark 19.  
**CAUTION:** Do not use mode=i8 (64 bit integer, 64 bit float) option when submitting the Nastran job if the ADAMSMNF EXPORT option specifies DB or BOTH and your intent is to use the DB in ADAMS. ADAMS does not currently support i8 DB files. If you use mode=i8, you must use the MNF in ADAMS.

NASTRAN/ADAMS MNF can be used with SOL 400 as described in Remark 21.

| Descriptor | Meaning  |
|------------|--|
| FLEXBODY   | Requests that the Nastran/ADAMS interface be executed. |
| NO         | Executes standard Nastran.                             |
| YES        | Executes Nastran/ADAMS interface.                      |

| Descriptor | Meaning   |
|------------|---|
| FLEXONLY   | Requests standard DMAP solution and data recovery following Nastran/Adams interface execution.  |
| YES        | Executes only the Nastran/ADAMS interface.  |
| NO         | Executes Nastran/ADAMS interface and standard DMAP solution and data recovery.                  |
| ADMCHECK   | Requests Nastran/ADAMS diagnostic output.   |
| YES        | Prints diagnostic output.   |
| NO         | Suppresses diagnostic output.   |
| ADMOUT     | Requests that the Nastran/ADAMS interface outputs Nastran .op2 files. .op2 files are generated. |
| YES        | Requests that .op2 files are not generated.   |
| NO         | Controls grid point stress output to .op2 file or .mnf or both.                                 |
| YES        | Grid point stress is output to .op2 file or .mnf or both.                                       |
| NO         | Grid point stress is not output to .op2 file, or .mnf file.                                     |
| OUTGSTRS   | Controls grid point strain output to .op2 file, or .mnf or both.                                |
| YES        | Grid point strain is output to .op2 file or .mnf or both.                                       |
| NO         | Grid point strain is not output to .op2 file or .mnf.   |
| OUTGSTRN   | Controls element stress output to .op2 file.  |
| YES        | Element stress is output to .op2 file.  |
| NO         | Element stress is not output to .op2 file.  |
| OUTSTRS    | Controls element strain output to .op2 file.  |
| YES        | Element strain is output to .op2 file.  |
| NO         | Element strain is not output to .op2 file.  |
| OUTSTRN    | Lower frequency bound of the Craig-Bampton modes in cycles/unit time.                           |
| value1     | Value of lower bound.   |
| V2ORTHO    | Higher frequency bound of the Craig-Bampton modes in cycles/unit time.                          |
| value2     | Value of higher bound.  |
| MINVAR     | Requests the type of mass invariants to be computed. See Remark 3.                              |
| FULL       | All nine mass invariants will be calculated.  |
| CONSTANT   | Only mass invariants (5-1), (5-2), (5-6), and (5-7) will be calculated.                         |
| PARTIAL    | All mass invariants except (5-5) and (5-9) will be calculated.                                  |
| NONE       | No mass invariants are computed.  |

| Descriptor                     | Meaning   |
|--------------------------------|---|
| RIGID                          | No modal information is output to the .mnf file. Only units, grid point coordinates, element connectivity, interface nodes, and invariant (5-1), (5-2), and (5-7) data are shared in the .mnf file. |
| EXPORT                         | Controls modal output.  |
| MNF                            | output to MNF file  |
| DB                             | output to Nastran database  |
| BOTH                           | output to Nastran database and MNF file   |
| PSETID                         | Selects a set of elements (including PLOTEL) whose grids are retained in the MNF, and whose connectivity defines face geometry for ADAMS display.   |
| <i>setid<sub>pplotel</sub></i> | Specified in the OUTPUT(PLOT) Section of Nastran.   |
| ALL                            | Select all the sets defined in the OUTPUT(PLOT) Section of Nastran.   |
| MONITOR                        | Request or suppress output of monitor point data to Adams MNF. Default is output (Yes). Specify No to suppress output.  |

**Remarks:**

1. This entry represents a collection of PARAM,name,value entries. A license is required for the Nastran/ADAMS interface.
2. ADAMSMNF FLEXBODY = YES is required to execute the Nastran/ADAMS interface, all other ADAMSMNF items are optional. The ADAMSMNF FLEXBODY = YES must occur above subcase level.  
If you want standard Nastran data recovery in addition to that produced for ADAMS, you need to run with FLEXONLY=NO. The orthonormal modes produced for ADAMS and the modes produced by the standard MSC Nastran run may be viewed in Patran if run with PARAM,POST,0.
3. The nine mass invariants are defined by Eqs. (5-1) through (5-9):

$$\begin{matrix} {}^1I \\ {}_{1 \times 1} \end{matrix} = \sum_{p=1}^N m_p \quad (5-1)$$

$$\begin{matrix} {}^2I \\ {}_{3 \times 1} \end{matrix} = \sum_{p=1}^N m_p s_p \quad (5-2)$$

$$\begin{matrix} {}^3I_j \\ {}_{3 \times M} \end{matrix} = \sum_{p=1}^N m_p \Phi_p \quad j = 1, \dots, M \quad (5-3)$$

$$\begin{matrix} {}^4I \\ {}_{3 \times M} \end{matrix} = \sum_{p=1}^N m_p \tilde{s}_p \Phi_p + I_p \Phi_p^* \quad (5-4)$$

$$\begin{matrix} {}^5 I_j \\ 3 \times M \end{matrix} = \sum_{p=1}^N m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1, \dots, M \quad (5-5)$$

$$\begin{matrix} {}^6 I \\ M \times M \end{matrix} = \sum_{p=1}^N m_p \Phi_p^T \Phi_p + \Phi_p^{*T} \mathbf{I}_p \Phi_p^* \quad (5-6)$$

$$\begin{matrix} {}^7 I \\ 3 \times 3 \end{matrix} = \sum_{p=1}^N m_p \tilde{s}_p^T \tilde{s}_p + \mathbf{I}_p \quad (5-7)$$

$$\begin{matrix} {}^8 I_j \\ 3 \times 3 \end{matrix} = \sum_{p=1}^N m_p \tilde{s}_p \tilde{\phi}_{pj} \quad j = 1, \dots, M \quad (5-8)$$

$$\begin{matrix} {}^9 I_{jk} \\ 3 \times 3 \end{matrix} = \sum_{p=1}^N m_p \tilde{\phi}_{pj} \tilde{\phi}_{pk} \quad j, k = 1, \dots, M \quad (5-9)$$

where  $s_p = [xyz]^T$  are the coordinates of grid point  $p$  in the basic coordinate system;

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix}$$

is the skew-symmetric vector cross product operator;  $\Phi_p$  is the partitioned orthogonal modal matrix that corresponds to the translational degrees-of-freedom of grid  $p$ ;  $\mathbf{I}_p$  is the inertia tensor;  $\Phi_p^*$  is the partitioned orthogonal modal matrix that corresponds to the rotational degrees-of-freedom of grid  $p$ ;  $\tilde{\phi}_{pf}$  is the skew-symmetric matrix formed for each grid translational degree-of-freedom for each mode;  $M$  is the number of modes; and  $N$  is the number of grid points.

4. The preceding mass invariant calculation currently depends on a lumped mass formulation. The parameter PARAM,COUPMASS should not be specified when executing the Nastran/ADAMS interface. Since p-elements use a coupled mass formulation, they should not be used.
5. If the CONM1 is used, M21, M31, and M32 entries should be left blank.
6. If PARAM,GRDPNT,value specified, mass invariants  ${}^1 I$ ,  ${}^2 I$ , and  ${}^7 I$  will be obtained from an Nastran grid point weight generator execution in the basic system.
7. The following DTI,UNITS Bulk Data entry is required for a FLEXBODY=YES run:

Since ADAMS is not a unitless code (as is Nastran), units must be specified. A DTI Bulk Data entry provides 'UNITS' (a unique identifier) input as the following example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed in the following table.

Format:

DTI UNITS 1 MASS FORCE LENGTH TIME

Example:

|     |       |   |    |   |   |   |
|-----|-------|---|----|---|---|---|
| DTI | UNITS | 1 | KG | N | M | S |
|-----|-------|---|----|---|---|---|

Mass:

- kg - kilogram
- lbf - pound-mass
- slug - slug
- gram - gram
- ozm - ounce-mass
- klbm - kilo pound-mass (1000.lbm)
- mgg - megagram
- sling - 12 slugs
- ug - microgram
- ng - nanogram
- uston - US ton

Length:

- km - kilometer
- m - meter
- cm - centimeter
- mm - millimeter
- mi - mile
- ft - foot
- in - inch
- um - micrometer
- nm - nanometer
- ang - angstrom
- yd - yard
- mil - milli-inch
- uin - micro-inch

Force:

- n - newton
- lbf - pounds-force
- kgf - kilograms-force
- ozf - ounce-force
- dyne - dyne
- kn - kilonewton
- klbf - kilo pound-force (1000.lbf)
- mn - millineutron
- un - micronewton
- nn - nanonewton

Time:

- h - hour
- min - minute
- s - sec
- ms - millisecond
- us - microsecond
- nanosec - nanosecond
- d - day

A note of clarification about UNITS and its relation to Nastran's WTMASS parameter: WTMASS, though necessary to achieve units consistency in Nastran, is ignored in the output for ADAMS. Units data for ADAMS is supplied on the UNITS DTI entry. For example, consider a model with mass in grams, force in Newtons, length in meters, and time in seconds. A WTMASS parameter equal to 0.001 would ensure that Nastran works with a consistent set of units (kg, N, and m). The units reported to ADAMS should then be: "DTI, UNITS, 1, GRAM, N, M, S."

8. OUTSTRS or OUTSTRN entries require the use of the standard Nastran STRESS= or STRAIN= Case Control commands to produce element stress or strain. STRESS(PLOT)= or STRAIN(PLOT)= will suppress stress or strain output to the Nastran .f06 file. The OUTSTRS or OUTSTRN entries are required for importing ADAMS results into MSC Fatigue. See the Nastran/ADAMS/durability documentation for more information.
9. OUTGSTRS or OUTGSTRN entries require the use of the standard Nastran STRESS= or STRAIN= Case Control commands used in conjunction with GPSTRESS= or GPSTRAIN= Case Control commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain output to the Nastran .f06 file.
10. To reduce the finite element mesh detail for dynamic simulations, PSETID=set\_entry is used to define a set of PLOTELS or other elements used to display the component in ADAMS. If a mass invariant computation is requested, this option can significantly reduce the size of the .mnf without compromising accuracy in the ADAMS simulation.

If PSETID specifies an existing set from the OUTPUT(PLOT) Section of Nastran, this single set is used explicitly to define elements to display in ADAMS. Otherwise, the Nastran Case Control Section will be searched for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements sets, the union of which will be used to define a set of PLOTELS or other elements used to display the component in ADAMS. If the user wishes to select all of the sets in the OUTPUT(PLOT) Section, then use PSETID=ALL.

The elements defined may include rigid element IDs. When defining these sets, do not use EXCLUDE and EXCEPT descriptions.

If a superelement analysis is being executed, any element defined on the PSETID=set\_entry that lies entirely on the superelement boundary (i.e., all of its grids are a-set or exterior to the superelement) must also be specified on a SEELT Bulk Data entry. The SEELT entry would not be required for part superelements, as boundary elements stay with their component.

```
OUTPUT (PLOT)
SET 7722 = 10001 THRU 10010
```

11. The ADMOUT=YES option is intended for users who plan to import ADAMS results into MSC Fatigue. This option requires the following assignment command in the File Management Section of the Nastran file:

```
ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM
```

It causes .op2 files with an .out extension to be generated for input into MSC Fatigue.

FLEXBODY=YES is required with its use. The files' outputs are: DTI-units, SE-number of superelements (9999 if residual), SEIDX-superelement id, ASETX-size of a-set, BGPDTS-grid location table, GEOM2S-element connections, GEOM4S-constraints and sets, MGGEW-physical mass external sort with weight mass removed, VAEXT-a-set partition vector, VGEXT-g-set partition vector, VAPEXT-eigenvalue size partition vector, MAAEW-modal mass, KAAE-modal stiffness, BAAE-modal damping, RAE-modal preload, PAE-modal loads, CMODEXT-component modes, OES1-element stress shapes, OSTR1-element strain shapes, OGS1-grid point stress shapes, OGSTR1-grid point strain shapes, OGSIPL-grid point physical preload stress, OGTRIPL-grid point physical preload strain. The files are output for each superelement and their generation depends on the loading and output requests.

To ensure compatibility with the ADAMS .op2-to-.mnf translator, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any .op2 files generated will have a pre-MSC Nastran 2004 format.

12. Environment variables controlling .mnf generation can be set before submitting the Nastran job, or by using the Nastran keyword 'MNFWRITEOPTIONS' which can be abbreviated to any short unique string such as 'MNFW'. The Nastran keyword can be entered on the NASTRAN submittal command line or in a user .nastran rc file. For example, interior grids and elements can be removed in the .mnf by entering, prior to the Nastran submittal, with Korn shell:

```
export MDI_MNFWRITE_OPTIONS=strip_face
```

with C shell:

```
setenv MDI_MNFWRITE_OPTIONS=strip_face
```

Or, at the time of Nastran submittal:

```
nastran_submittal_command jid MNFW=strip_face
```

Consult the ADAMS/Flex documentation for more information on the use of environment variables during .mnf generation.

The command

```
nastran_submittal_command help mnfw
```

will generate the description of the symbol keyword.

The command

```
nastran_submittal_command help all
```

will generate a complete set of Nastran submittal keywords.

13. .mnf naming convention is as follows: for a single superelement run, 'jid.mnf'; for a residual only or multiple superelement run 'jid\_seid.mnf', etc.; where seid1 and seid2 are the integer numbers of the superelement. The default location of these files is the same directory as the jid.f06 file. See the ASSIGN .mnf command to change directory location.
14. When supplying SPOINT/QSET combinations, there should be a sufficient number of combinations to correctly capture the modal shapes. If  $n$  is the number of modes specified on the EIGR or EIGRL Bulk Data entries and  $p$  is the number of loadcases specified, then the number of SPOINTS( $ns$ ) should be at least  $ns = n + (6 + p)$  assuming that residual flexibility is on. In general, there cannot be too many SPOINTS, as excess ones will simply be truncated with no performance penalty.
15. The user can have Nastran automatically specify the SPOINT/QSET by including, above the Case Control Section the parameter PARAM,AUTOQSET,YES. In this case no SPOINT/QSET can appear in the Bulk Data. See the PARAM,AUTOQSET description for detailed requirements or limitations.

16. By default, MSC Nastran 2005 will create a version 6.0 MNF. ADAMS 2005 is able to read the version 6.0 .mnf file. Earlier ADAMS versions are not able to read a 6.0 .mnf file. Nastran can be instructed to write a backward-compatible .mnf file by submitting the Nastran job with MNFWRITEOPTIONS=full\_str. Alternatively, the user may set the environment variable MDI\_MNFWRITE\_OPTIONS to 'full\_str'. See Remark 12. for more information on controlling the .mnf format.
17. In addition to modal stiffness and modal mass matrices, the modal damping matrix may also be output to the .mnf. The damping allowed is the standard Nastran damping matrix consisting of  $[B_{gg}]$  viscous damping,  $(1/w4) [K^4_{gg}]$  structural damping,  $(g/(w3) + \alpha2)[K_{gg}]$  structural and Rayleigh damping, and  $(\alpha1)[M_{gg}]$  Rayleigh damping. Where g is set by PARAM,G,value, w3 is set by PARAM,W3,value; w4 is set by PARAM,W4,value;  $\alpha1$  is set by PARAM,ALPHA1,value,0.; and  $\alpha2$  is set by PARAM,ALPHA2,value,0.

Additionally,  $[B2H]$  modal damping can be included by use of the Case Control command SDAMP=n. For part superelement or superelement analyses, modal damping for each individual part or superelement can be controlled by PARAM,SESDAMP,YES (PARAM,SESDAMP,NO is the default).

| SESDAMP  |   |   |
|--|---|---|
| sesdamp = no   | Modal damping for each superelement using the free boundary modes.      | $SDAMP \Rightarrow TABDMP1$<br>SDAMP above subcase.           |
| sesdamp = yes  | Modal damping for each superelement using the fixed boundary CMS modes. | $SDAMP \Rightarrow TABDMP1$<br>SDAMP in superelement subcase. |
| For part superelements<br>each part may have a PARAM,SESDAMP |   |   |

The Nastran/ADAMS interface does not allow for adding modal damping to structural damping using PARAM,KDAMP,-1.

Direct input damping may also be included with the Case Control command B2GG=n. For part superelement or superelement analyses, use of this command with the Nastran/ADAMS interface requires fully expanded case control.

18. If preload is present in the model, physical gridpoint stress and strain for the preload may be output to the .mnf using standard GPSTRESS= or GPSTRAIN= commands.  
If preload is generated in a SOL 106 run for a SOL 103 restart, and the physical grid point stresses for the preload are desired for the SOL 103 MNF run, then PARAM,FLEXNLS,YES is required above the subcase level in the SOL 106 run.  
For preload generated in a SOL 106 run for a SOL 103 restart, the preload subcase must be replicated in the first subcase of the SOL 103 run.

19. After using the Nastran/ADAMS interface to produce an .mnf file and after performing an ADAMS solution, it is possible to bring the ADAMS results into Nastran for modal data recovery. ADAMS produces .op2 files for input to Nastran SOL 111 and SOL 112. The files are binary format with an .mdf extension. The File Management Section requires an assign command for each file:

```
ASSIGN INPUTT2='name.mdf' UNIT=ni
```

with a DLOAD = ni in the appropriate subcase.

Also, in the Bulk Data Section, the parameter PARAM,ADMPOST = m (m = 0, by default no MSR performed) is required. If m = 1, rigid body motion is not considered in the structural deformation. If m = 2, rigid body motion is considered in the structural deformation.

Full details of the generation of the .mdf files and their use with Nastran are to be found in the Nastran/ADAMS durability documentation.

20. In order to obtain consistent ADAMS results in Nastran data recovery run, when ADAMS data brought back into Nastran SOL 111 or SOL 112, Nastran must be restarted from either of the following two options:

- From the original Nastran database that was produced during SOL103 MNF creation run (created MNF is used in ADAMS run).
- From the external SE (\*.op2) that was produced during SOL103 MNF creation run (created MNF is used in ADAMS run). The File Management section requires to assign command for external SE op2.

```
ASSIGN INPUTT2='EXTSE100.OP2' UNIT=ni
```

with following PARAM entries

```
PARAM ADMPOST,1 $
```

```
PARAM ADMEXTU , ni $ Where ni = unit # of external SE op2.
```

Note that ADMPOST=2 is not supported with this restart option (via external SE op2).

21. Typically SOL 400 is used to produce a preload for an ADAMS flexbody MNF run.

- In the preload run the structure should be statically supported and follower loading must be applied as a self equilibrating load set (not with SPC relationships!). In the ANALYSIS=MODES step the structure must be a free-free structure as the resulting orthonormalization requires that six rigid body modes be present.

In order to produce modal amplitudes and mode shapes and to ensure residual vector calculations, SPOINTs and Q-sets are required. The SPOINTs must be included in the MAIN Bulk Data as they are included in the overall matrix size.

A new BULK data section labeled BEGIN FLXBDY =id must be included with the run. This new bulk section must contain the Q-set associate with the SPOINTs (in main bulk!) for modal amplitudes and the A-set required for attachment point designation.

The example below is a typical SOL400 problem setup:

```
SOL 400
CEND
$ Case Control Section
$ Output ADAMSMNF REQUIRED ABOVE SUBCASE
```

```

ADAMSMNF flexbody=yes, psetid=all, outgstrs=yes, outgstrn=yes
SUBCASE 1
    $ Preload
    STEP 10
        $ Static load and support for preload
        SUBTITLE = PRELOAD
        ANALYSIS = NLSTATICS
        NLSTEP = 110
        LOAD = 120
        SPC = 130
        BCONTACT = 140
        SPCF = ALL
        $ Generate stress and strain grid shapes
        STRESS(PLOT) = ALL
        STRAIN(PLOT) = ALL
        GPSTRESS(PLOT) = ALL
        GPSTRAIN(PLOT) = ALL
    $ Modal Step for Producing MNF
    $ Default: Select the end of previous load step to output
$ ADAMSMNF
    STEP 20
        ANALYSIS = MODES
        $ Select real Eigen Value Parameters
        METHOD = 210
        $ Turn residual vectors on
        RESVEC = COMPONENT
        STRESS(PLOT) = ALL
        STRAIN(PLOT) = ALL
        GPSTRESS(PLOT) = ALL
        GPSTRAIN(PLOT) = ALL

```

- b. In the above example, the SPC set in the ANALYSIS=NLSSTAT must be a static (non-redundant) constraint condition. Note that in the ANALYSIS=MODES STEP, the SPC constraint set has been removed.

In SOL 400, the definition of the attachment a-set for identifying attachment points and for q-set for specifying the desired number of modal amplitudes for orthonormalization is done in a separate new FLXBDY Bulk Data Section shown below:

```

$ FLEXBODY Bulk section
BEGIN BULK FLXBDY = 10
$ Attachment point and component mode (A-SET) selection
ASET1,123456,1,11,111,121
QSET1,0,100001,THRU,100020

```

In SOL400, the ASET/ASET1 and QSET/QSET1, MUST, appear in the FLXBDY bulk data section.

Currently, only one FLXBDY bulk data section (with a positive Flexbody ID) is supported in SOL 400. Any non set related Bulk entries appearing in this Section will be ignored.

- c. If CONTACT is required as part of the preloading for the FLEXBODY=YES run, it is Highly Recommended that the friction option be turned on by using an appropriate BCPARA bulk data entry setting, e.g.,

```
$ Select bilinear Coulomb friction for all subcases  
BCPARA, 0, FTTYPE, 6
```

If contact friction is not turned on, the tangential motion between the two parts coming into contact will most likely not be constrained and incorrect or fatal results will occur.

- d. If RIGID elements (RBE1/RBE2/RBE3/RBAR/RROD/RJOINT) are in the model, then the Case Control RIGID= LAGRANGE (default for SOL400) should be used to avoid possible wrong results. If an attachment point happens to touch a rigid element, the point should be associated with the independent degree of freedom of the rigid element. Though not recommended, if for some modeling requirement, a dependent rigid element grid is required to be in the attachment set, the user MUST include at least one independent/reference grid for that specific rigid element in the ASET.
22. Real coupled method does not support in ADAMS interface. If Case Control METHOD(COUPLED) is used with ADAMSMNF FLEXBODY=yes, the following error message will be returned:  
**\*USER FATAL MESSAGE 22985 (SUBDMAP IFPL)  
CASE CONTROL COMMAND METHOD(COUPLED) IS NOT SUPPORTED WITH THE  
NASTRAN/ADAMS INTERFACE.**

**ADAPT****p-Adaptivity Control Selection**

Specifies p-adaptivity control parameters.

**Format:**

ADAPT=n

**Example:**

ADAPT=12

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification for either an ADAPT or PSET Bulk Data entry (Integer > 0). |

**Remarks:**

1. ADAPT is required only when an analysis with p-elements is requested.
2. A multiple p-level analysis with error analysis is performed whenever the ADAPT command references an ADAPT Bulk Data entry.
3. A single p-level analysis without error analysis is performed whenever the ADAPT command references a PSET Bulk Data entry.
4. Only one ADAPT may appear in the Case Control Section, and should appear above all SUBCASE commands.
5. The subcases that will not participate in the error analysis/p-adaptivity must contain the ADACT=NONE command.

**AECONFIG**

## Aeroelastic Configuration Name

Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.

Format: Assign a DBset member name

AECONFIG =config-name

Example:

Assign a MASTER file for the aerodynamic and aeroelastic DBsets

AECONFIG =PROTO\_A

| Describer   | Meaning  |
|-------------|--|
| config-name | The configuration name. This is the aerodynamic supergroup name identified as part of the aeroelastic model (Character; Default = AEROSG2D). |

Remarks:

1. Typically, the aeroelastic configuration name is specified as the aerodynamic supergroup as part of the model generation in MSC FlightLoads.
2. If AECONFIG is not present, aerodynamic and aeroelastic datablocks will be created from the data in the Bulk Data Section and assigned the default value AECONFIG=AEROSG2D.
3. Multiple configuration names are supported.
4. AECONFIG is typically assigned above the subcase level. If it is overridden at the subcase level, it is necessary to attach an existing aerodynamic database.

**AERCONFIG****Aerodynamic Model to be Used for the Rigid Aerodynamics**

Enables the user to select a different mesh for the rigid portion of the aerodynamics than for the elastic portion.

**Format:**

AERCONFIG=config-name

**Examples:**

AERC=RAERO

| Descriptor | Meaning  |
|------------|--|
| AERC       | The configuration name. This is the aerodynamic supergroup name for the aerodynamic model that is used to create the rigid aerodynamics. |

**Remarks:**

1. If the AERCONFIG Case Control command is not present in the subcase, the rigid portion of the aerodynamics is based on the same AECONFIG as the flexible aerodynamics.
2. The rigid aerodynamics must be precomputed and attached from an assigned database using FMS commands such as:

```
ASSIGN RMASTER = "raero.master"  
DBLOCATE WHERE(AECONFIG='rconfig') LOGICAL=RMASTER
```

**AEROF****Aerodynamic Force Output Request**

Requests the aerodynamic loads on aerodynamic control points.

**Format:**

AEROF=n

**Examples:**

AEROF=ALL

AEROF=5

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a previously appearing SET command (Integer > 0). |
| ALL        | Forces at all points will be output.                                    |

**Remarks:**

1. This command is supported in SOLs 144, 146 (frequency response only) and 200 for ANALYSIS=SAERO.
2. The SET command references box or body element identification numbers.
3. Output is in the units of force or moment.
4. Only aerodynamic forces on points specified on the SET command will be output.

**AESYMXY****Aerodynamic Flow Symmetry About The XY Plane**

Aerodynamic XY plane of symmetry flag. This is used to indicate whether the aerodynamic model has symmetry with respect to the ground.

**Format:**

$$\text{AESYMXY} = \begin{Bmatrix} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{Bmatrix}$$

**Example:**

AESYMXY = ASYMMETRIC

| Descriptor    | Meaning   |
|---------------|---|
| SYMMETRIC     | Indicates that the aerodynamic model is moving in a symmetric manner with respect to the XY plane.      |
| ANTISYMMETRIC | Indicates that the aerodynamic model is moving in an antisymmetric manner with respect to the XY plane. |
| ASYMMETRIC    | Indicates that the aerodynamic model has no reflection about the XY plane.                              |

**Remarks:**

1. If AESYMXY is not present in case control, aerodynamic XY symmetry will be determined from the SYMXY field of the AEROS Bulk Data entry for static aeroelastic analysis, and from the SYMXY field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXY is present above the subcase level, it is applied to all subcases until overridden.
3. SYMMETRIC implies ground effect, and ASYMMETRIC implies free air analysis.
4. Multiple aerodynamic symmetries are supported.

**AESYMXZ**

## Aerodynamic Flow Symmetry About The XZ Plane

Aerodynamic XZ plane of symmetry flag. This is used to support symmetric models about the centerline.

Format:

$$\text{AESYMXZ} = \left\{ \begin{array}{l} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{array} \right\}$$

Example:

`AESYMXZ = SYMMETRIC`

| Descriptor    | Meaning   |
|---------------|---|
| SYMMETRIC     | Indicates that a half span aerodynamic model is moving in a symmetric manner with respect to the XZ plane.      |
| ANTISYMMETRIC | Indicates that a half span aerodynamic model is moving in an antisymmetric manner with respect to the XZ plane. |
| ASYMMETRIC    | Indicates that a full aerodynamic model is provided (Default).  |

Remark:

1. If AESYMXZ is not present in case control, aerodynamic XZ symmetry will be determined from the SYMXZ field of the AEROS Bulk Data entry for static aeroelastic analysis, and from the SYMXZ field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXZ is present above the subcase level, it is applied to all subcases until overridden.
3. Multiple aerodynamic symmetries are supported.

**AEUXREF**

Defines the Reference Aerodynamic Extra Point (Controller) Vector

Reference UXVEC selector for the aeroelastic trim analysis. This is used to indicate an aerodynamic extra point vector about which the stability derivatives are to be computed and printed. The stability derivatives are the change in force due to a unit perturbation of each parameter in the aerodynamic extra point set. Due to the nonlinear nature of the aeroelastic loads, the stability derivatives can be (but are not required to be) a function of the point about which the slope is computed. This input defines which point is to be used in computing the stability derivatives for printing (local slopes will be computed as needed in the trim solver). This selection is typically done within each subcase, but a case control default can be defined by placing an entry above the subcase level.

**Format:**

$$\text{AEUXREF} = \left\{ \begin{matrix} n \\ \text{TRIM} \end{matrix} \right\}$$

**Examples:**

AEUXREF=100

AEUXREF=TRIM

| Descriptor | Meaning   |
|------------|---|
| n          | The identification number of a UXVEC Bulk Data entry that defines the point about which stability derivatives will be computed in TRIM cases. |
| TRIM       | Indicates that the stability derivatives should be computed about the trimmed state.  |

**Remarks:**

1. If, for a particular subcase, AEUXREF is not defined, the “free stream” state will be used (that is, the stability derivatives will be computed about zero values for all parameters). This results in upward compatibility with the linear database paradigm.
2. Only one of a TRIM or a UXVEC ID may be specified on any given subcase. To see stability derivatives about additional points, you must define additional trim subcases.

**ANALYSIS****Analysis Discipline SUBCASE/STEP/SUBSTEP Assignment**

Specifies the type of analysis being performed for the current SUBCASE/STEP/SUBSTEP.

**Format:**

ANALYSIS=type

**Examples:**

SOL 200

```
SUBCASE 10
  ANALYSIS=STATIC
SUBCASE 20
  ANALYSIS=MODES
```

| Descriptor | Meaning   |
|------------|---|
| type       | Analysis type. Allowable values and applicable solution sequences (Character):  |
| STATICS    | Linear Static Analysis (SOLs 200 & 400)   |
| MODES      | Normal Modes Analysis (SOLs 110, 111, 112, 106, 200, & 400) See Remarks <a href="#">2</a> , <a href="#">3</a> , and <a href="#">4</a> . |
| BUCK       | Buckling (SOLs 200 and 400)   |
| DFREQ      | Direct Frequency (SOLs 106, 200 and 400). See Remark <a href="#">4</a> .  |
| MFREQ      | Modal Frequency (SOLs 200 and 400)  |
| MTRAN      | Modal Transient (SOLs 200 and 400)  |
| DCEIG      | Direct Complex - Eigenvalue Analysis (SOLs 200 & 400)   |
| MCEIG      | Modal Complex - Eigenvalue Analysis (SOLs 200 & 400)  |
| SAERO      | Static Aeroelasticity (SOLs 200 and 400)  |
| DIVERGE    | Static Aeroelastic - Divergence (SOLs 200 & 400).   |
| FLUTTER    | Flutter (SOLs 200 and 400)  |
| HEAT       | Heat Transfer Analysis (SOLs 153, 159 and 600 only)   |
| STRUCTURE  | Structural Analysis (SOLs 153 & 159 only, default for these two SOLs)   |
| HSTAT      | Steady State Heat Transfer (SOL 400)  |
| HTRAN      | Transient Heat Transfer (SOL 400)   |
| NLSTATICS  | Nonlinear Static Analysis (SOL 400). See Remark   |
| NLTRAN     | Nonlinear Transient Analysis (SOL 400). See Remark <a href="#">3</a> .  |
| HOT2COLD   | Hot-to-cold Analysis (SOL 106 only). See Remark <a href="#">4</a> .   |

| Descriptor | Meaning                                |
|------------|--|
| RCNS       | RC Network Thermal Solver Steady State |
| RCNT       | RC Network Thermal Solver Transient    |

### Remarks:

1. This entry is used in solution sequences that are capable of multiple analysis types and selects the actual analysis to be performed in the SUBCASE (or for SOL 400 the SUBCASE, STEP, or SUBSTEP).
  - For linear solution sequences SOLs 110, 111, and 112 and the nonlinear SOLs 106, 153, and 159, and for Optimization SOL 200 there can be only one **ANALYSIS=** per SUBCASE
  - SOL 600,153 and SOL 600,159 require **ANALYSIS=HEAT**. For other SOL 600, ID executive control entries, **ANALYSIS=** is not used and should not be entered.
  - For SOL 200, EVERY SUBCASE including any superelement SUBCASE must be assigned an **ANALYSIS=** Case Control command either in the explicit SUBCASE or above all SUBCASEs in which case it defaults to all the subcases.
  - For SOL 400 there can, for single physics analysis or chained multi physics analysis, be only one **ANALYSIS=** per STEP.
  - For SOL 400 coupled multi physics analysis there can be only one **ANALYSIS=** per SUBSTEP.
2. In the linear solution sequences SOLs 110, 111, 112 this entry allows data recovery of the normal modes data used in the complex Eigenvalue analysis, modal frequency analysis, or modal transient analysis. All commands which control the boundary conditions (SPC, MPC, and SUPPORT) and METHOD selection should be copied inside the **ANALYSIS=MODES** SUBCASE or specified above the SUBCASE level. Note in the example below, SOL111 is a frequency response solution sequence so in the frequency response SUBCASE 2 below an **ANALYSIS=MFREQ** is allowed but not required.

```

SOL 111
METH=40
SPC=1
SUBCASE 1 $ Normal Modes
  ANALYSIS=MODES
  DISP=ALL
SUBCASE 2 $ Frequency response
  STRESS=ALL
  DLOAD=12
  FREQ=4

```

3. SOL 400 analysis allows for seven analysis type combinations (nonlinear single physics, nonlinear chained physics, nonlinear coupled physics, linear perturbation analysis, standard linear physics, nonlinear chained analysis with mesh/time change physics, and nonlinear response optimization with ESLNRO (Equivalent Static Loads Nonlinear Response Optimization)).

If there are linear, nonlinear and perturbation subcases, the linear subcases will be solved first. The linear subcases are reordered for processing and the output will be in the following order regardless of original subcase number: STATICS, MODES, BUCKLING, DFREQ or MFREQ, DCEIG or MCEIG, SAERO, FLUTTER.

The general rule is: The solutions of all SUBCASEs are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP in the same SUBCASE. The solutions of the SUBSTEPs occur simultaneously within a STEP (coupled analysis):

Additionally the following rules should be observed:

- All Coupled Multi-Physics steps have to come before the Single-Physics steps.
  - Single-Physics steps can follow the Multi-Physics Steps.
  - All linear perturbation steps need to be at the end after definition of all possible coupled multi-physics steps and single-physics steps.
  - The Case Control Command NLIC used for Nonlinear Initial Condition should be referenced for further requirements between the analysis types allowed in SOL 400.
- a. Nonlinear single physics: Nonlinear structures or nonlinear heat: For nonlinear structures a **ANALYSIS=NLSTAT** must come before a **ANALYSIS=NLTRAN**. Only Statics to Transient is allowed for structures. For heat transfer a nonlinear steady state heat transfer to a nonlinear transient heat transfer is not allowed.

A structural example:

```
SUBCASE 1
  STEP 10
    ANALYSIS=NLSTAT
  STEP 20
    ANALYSIS=NLSTAT
  STEP 30
    ANALYSIS=NLTRAN
```

- b. Nonlinear chained physics: A nonlinear steady state heat with results used for a nonlinear static structural analysis. Only a steady state heat to a structural nonlinear Statics is allowed.

```
SUBCASE 1
  STEP 10
    ANALYSIS=HSTAT
  STEP 20
    ANALYSIS=NLSTAT
```

- c. Nonlinear coupled physics: This allows for four combinations of STEP/SUBSTEP within the SUBCASE
- **ANALYSIS=HSTAT** for the first SUBSTEP and **ANALYSIS=NLSTAT** for the second SUBSTEP
  - **ANALYSIS=HTRAN** for the first SUBSTEP and **ANALYSIS=NLTRAN** for the second SUBSTEP
  - **ANALYSIS=HTRAN** for the first SUBSTEP and **ANALYSIS=NLSTAT** for the second SUBSTEP

- **ANALYSIS=HSTAT** for the first SUBSTEP and **ANALYSIS=NLTRAN** for the second SUBSTEP

A coupled analysis example is:

```
SUBCASE 100
  STEP 10
    STRESS= ALL
    NLSTRESS=ALL
    NLSTEP=84
  SUBSTEP 1
    ANALYSIS=HSTAT
    THERMAL=ALL
    FLUX=ALL
    SPC=35
    LOAD=11
  ANALYSIS=NLSTAT
    SPC=2
    LOAD=110
    DISP(PLOT)=1456
  STEP 20
    ANALYSIS=NLTRAN (single physics rules follow)
```

- d. Linear perturbation analysis: Linear perturbation analysis is run directly after a nonlinear static (ANALYSIS=NLSTAT) analysis using additional STEP commands containing ANALYSIS=BUCK, MODES, DFREQ, MFREQ, MTRAN, DCEIG, and MCEIG entries.

It should be noted that ANALYSIS=BUCK for linear perturbation is not recommended if the model is highly nonlinear. NLBUCK Case Control command is recommended for highly nonlinear models.

An example is:

```
SOL 400
CEND
TITLE=MSC Nastran SOL 400, Linear Perturbation Analysis
SUBTI=3D General Contact with Large Displacement Turned on
$
SUBCASE 1
  STEP 1
    LABEL=Nonlinear Static Analysis with Contact
    ANALYSIS = NLSTATIC
    NLPARM = 1
    BCONTACT = 1
    BOUTPUT=ALL
    SPC = 2
    LOAD = 3
    DISPLACEMENT(SORT1,REAL)=ALL
STEP 2
  LABEL=Linear Perturbation, DFREQ
  ANALYSIS = DFREQ
  DLOAD=200
  FREQ =10
  AUTOSPC=YES
  SPC = 2
  DISPLACEMENT = ALL
```

```

STEP 3
  LABEL=Linear Perturbation, MFREQ
  ANALYSIS = MFREQ
    NLIC STEP 1 LOADFAC 1.0
    METHOD = 30
    DLOAD=200
    FREQ =10
    AUTOSPC=YES
    RESVEC =NO
    SPC = 2
    DISPLACEMENT = ALL

```

- e. Standard linear physics: Runs can include ANALYSIS=STATICS, MODES, BUCKL, DFREQ, MFREQ, MTRAN, DCEIG, MCEIG, SAERO, and FLUTTER. These are standard SUBCASE type of analysis.

An example is:

```

SUBCASE 2
  LABEL=Linear Static
  ANALYSIS = STATIC
    LOAD      = 1001
    DISP      = 10
    STRESS    = ALL
SUBCASE 101
  DISP      = 10
  STRESS    = ALL
  $
STEP 11
  LABEL=Nonlinear Statics, Load 1001
  ANALYSIS = NLSTATIC
    NLPARM   = 11
    LOAD     = 1001
  $
STEP 12
  LABEL=Nonlinear Statics, Load 1005
  ANALYSIS = NLSTATIC
    NLPARM   = 11
    LOAD     = 1005
  $
STEP 13
  LABEL=Linear Perturbation, Modes
  ANALYSIS = MODES
    METHOD   = 1003
    RESVEC   = NO
    AUTOSPC(NOPRINT) = YES
    DISPL    = ALL
SUBCASE 1004
  LABEL=Get Linear Normal Modes
  ANALYSIS = MODES
    SVECTOR  = ALL
    METHOD   = 1004

```



- f. Nonlinear chained analysis with mesh/time change physics: A standard single physics nonlinear steady state ANALYSIS=HSTAT or transient heat transfer ANALYSIS=HTTRAN with either scratch=no or scratch=mini on the job submittal. This is followed by a mechanical job submittal with an ASSIGN hrun='name\_of\_heat\_run.MASTER' and DBLOC DATABLK=(HEATDB) LOGI=hrun in the File Management Section of the Nastran executive and an ANALYSIS=NLSSTAT or ANALYSIS=NLTTRAN with a TEMP(LOAD,HSUBC,HSTEP,HTIME) in the subsequent mechanical job STEP. The subsequent mechanical job can have both a different mesh than the heat job and different time steps.

An example:

```
SOL 400 (submitted with SCRATCH=MINI from a bulk file named
Course_Mesh_Heat.dat)
CEND
SPC = 1
IC = 10
THERMAL=ALL
SUBCASE 3
    STEP 4
        ANALYSIS=HSTAT
        NLPARM = 1
        LOAD = 202
    SUBCASE 10      (Note if STEP not provided STEP=1 defaulted)
        ANALYSIS=HTTRAN
        = 2
        DLOAD = 404
BEGIN BULK

SOL 400      (Mechanical run using solution from
Course_Mesh_Heat.dat)
ASSIGN hrun= 'Course_Mesh_Heat.MASTER'   (the ticks ( ' ) are
required)
DBLOC DATABLK=(HEATDB),    LOGI=hrun
CEND
TEMPERATURE(INITIAL) = 1
SUBCASE 1
    ANALYSIS=NLTTRAN
    STEP 1
    = 1
    SPC = 2
    TEMP (LOAD, HSUBC=3) = 3
    DISPLACEMENT(SORT1,REAL)=ALL
    NLSTRESS = ALL
    STRESS = ALL
    STEP 2
    = 2
    SPC = 2
    TEMP (LOAD, HSUBC=10, HTIME=0.80) = 4
    DISPLACEMENT(SORT1,REAL)=ALL
    NLSTRESS = ALL
    STRESS = ALL
SUBCASE 2
    ANALYSIS=NLTTRAN
```



```

STEP 3
= 3
SPC = 2
TEMP (LOAD, HSUBC=10, HTIME=ALL) = 5
DISPLACEMENT(SORT1,REAL)=ALL
NLSTRESS = ALL
stress = all
BEGIN BULK

```

- g. Nonlinear response optimization with ESLNRO: A standard single physics nonlinear **ANALYSIS=NLSSTAT** is performed with a **NASTRAN ESLOPT=1** in the bulk data file. Besides the usual model grid, loads, element data etc., the bulk data file should contain the standard SOL 200 design criteria.

An example:

```

NASTRAN ESLOPT=1    $
SOL 400
CEND
DESOBJ(MIN) = 10000
  ANALYSIS = NLSTATIC
  DESSUB = 1
  LOAD = 300
  SPC = 1
  STRESS = ALL
  DISP = ALL
  NLPARM = 1
  BEGIN BULK

```

4. In SOL 106, the ANALYSIS Case Control command may be used to define a ‘linear’ perturbation analysis SUBCASE or a user input of the ‘stressed’ or deformed geometry (normal bulk data input) and ‘unload’ the structure to determine the unstressed shape SUBCASE.
- a. ‘Linear’ perturbation analysis: The ANALYSIS Case Control command may be used to define a ‘linear’ perturbation analysis SUBCASE, separate from the subcases used to load the model. Normal modes and frequency response subcases with **ANALYSIS=MODES** or **ANALYSIS=DFREQ** will use the final displacement results and loads from the previous nonlinear subcase to generate the stiffness, differential stiffness, and follower force matrices for use in the ‘linear’ response analyses. Data recovery will be based on the requests above and within the subcase.

An example is:

```

SUBCASE 1
$ LOAD STRUCTURE
LOAD= 100
NLPARM= 100
SUBCASE 2
$ NORMAL MODES
ANALYSIS= MODES
METHOD= 100
DISP= ALL
SUBCASE 3
$ FREQUENCY RESPONSE
ANALYSIS= DFREQ

```

```

SET 100= 1 THRU 1000
DISP= 100
SUBCASE 4
$ CONTINUE LOADING STRUCTURE
LOAD= 200
NLPARM= 100
DISP= ALL
SUBCASE 5
$ NORMAL MODES AT THE NEW LOADING
ANALYSIS= MODES

```

In the previous example, subcases 2 and 3 will use the results from subcase 1, subcase 4 will continue the loading application, and subcase 5 will use the results from subcase 4. For the ‘linear’ analyses, the mass matrix will be based on the undeformed geometry and the damping matrix will be generated using the deformed geometry. This will allow analyses of ‘large’ displacement results (PARAM, LGDISP, 1), in addition to material nonlinear analyses.

- User input of the ‘stressed’ or deformed geometry: **ANALYSIS=HOT2COLD** allows the user to input the ‘stressed’ or deformed geometry (normal bulk data input) and ‘unload’ the structure to determine the unstressed shape. This uses an iterative technique with each iteration a nonlinear analysis. This feature will also ‘reset’ the geometry to the ‘unstressed’ position for additional subcases. Related PARAMETERS are HTOCTOL, HTOCITS, and HTOCPRT.

An example is:

```

SUBCASE 1
$ UNLOAD STRUCTURE
ANALYSIS=HOT2COLD
LOAD= 100
NLPARM= 100
SUBCASE 2
$ NORMAL MODES UNLOADED STRUCTURE
ANALYSIS= MODES
METHOD= 100
DISP=ALL
SUBCASE 3
$ LOAD the STRUCTURE
LOAD= 100
NLPARM= 100
SUBCASE 4
$ NORMAL MODES OF LOADED STRUCTURE
ANALYSIS= MODES
METHOD= 100
DISP= ALL

```

SUBCASE 1 unloads the model. SUBCASE 2 calculates the modes of the ‘undeformed’ structure; the differential stiffness and follower force effects will not be included. SUBCASE 3 loads the model. SUBCASE 4 calculates the modes of the loaded structure; the differential stiffness and follower forces effects will be included.

**APPRESSURE****Aerodynamic Pressure Output Request**

Requests the aerodynamic pressures in static aeroelastic response.

Format:

$$APRES = \left\{ \begin{array}{l} n \\ ALL \end{array} \right\}$$

Examples:

APRES=ALL

APRES=6

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification number of a previously appearing SET command. Only aerodynamic pressures on the referenced aerodynamic boxes will be output (Integer > 0). |
| ALL        | Pressures at all points will be output.   |

**AUTOSPC**

Constrains Stiffness Singularities via m-sets or s-sets

Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.

Format:

$$\text{AUTOSPC} \left[ \left[ \begin{array}{l} [\text{RESIDUAL}] \\ \left[ \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right] \end{array} \right], \left[ \begin{array}{l} \left[ \begin{array}{l} \text{NOPUNCH} \\ \text{PUNCH} \end{array} \right] \\ \left[ \begin{array}{l} \text{SPC} \\ \text{MPC} \end{array} \right], \left[ \begin{array}{l} \text{ZERO} \\ \text{NOZERO} \end{array} \right] \end{array} \right] \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

Examples:

**AUTOSPC=YES**

**AUTOSPC (PRINT, PUNCH, SID=100, EPS=1.E-6, MPC)=YES**

| Descriptor | Meaning   |
|------------|---|
| RESIDUAL   | For SOL 400, applies AUTOSPC to both the residual structure and superelements. See Remarks 6. and 7.  |
| PRINT      | For SOL 101, applies AUTOSPC to linear analysis with contact. See Remark 9.   |
| NOPRINT    | Enables the printout of a summary table of singularities (Default).   |
| NOPUNCH    | Disables the printout of a summary table of singularities.  |
| PUNCH      | Disables the creation of SPC or MPC Bulk Data entries in the PUNCH file (Default).  |
| SID=n      | Generates SPC or MPC Bulk Data entry format in the PUNCH file.  |
| EPS=r1     | Specifies a set identification number for option PUNCH (Default = 999).   |
| EPSSING=r2 | Identifies singularities with a stiffness ratio smaller than r1 to be automatically constrained with single or multipoint constraints. See Remark 2. (Default = 1.E-8). |
| SPC        | Identifies the potential singularities with stiffness ratios less than r2. See Remark 2. (Default=1.E-8).   |
| MPC        | Applies single-point constraints on degrees of freedom identified as singular. (Default)  |
| ZERO       | Applies multipoint constraints on degrees of freedom identified as singular.  |
| NOZERO     | Requests the printout of singularities with zero stiffness ratios in the singularity summary table (Default).   |
|            | Disables the printout of those singularities with zero stiffness ratios in the singularity summary table.   |

## Remarks:

1. AUTOSPC specifies the action to take when singularities exist in the stiffness matrix. AUTOSPC=YES means that singularities will be constrained automatically. AUTOSPC=NO means that singularities will not be constrained. If AUTOSPC=NO, then the user should take extra caution analyzing the results of the grid point singularity table and the computed epsilons. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for details of singularity and mechanism identification and constraint.
2. Singularity ratios smaller than EPSSING are listed as potentially singular. If AUTOSPC=YES, then the identified singularities with a ratio smaller than EPS will be automatically constrained. If EPSSING has the same value as EPS, then all singularities are listed. If EPSSING is larger than EPS, the printout of singularity ratios equal to exactly zero is suppressed. EPSSING must be greater than or equal to EPS. If not, the program will set EPSSING equal to EPS.
3. If the PUNCH keyword is specified, then automatically generated SPCs or MPCs are placed in SPCi or MPCi Bulk Data entry format on the PUNCH file.
4. By default, in all solution sequences except 106, 129 and 400, the auto-SPC operation is performed for both superelements and the residual structure. (Auto-SPC processing is disabled in heat transfer analysis). In SOLs 106, 129 and 400, the default is to perform the operation only on superelements. If it is desired to perform auto-SPC on the residual structure in:
  - a. SOLs 106, then specify PARAM,AUTOSPCR,YES and is only applied to the omitted degrees-of-freedom. The AUTOSPC command is ignored for the residual structure.
  - b. SOLs 129, then specify PARAM,AUTOSPCR,YES. The AUTOSPC command is ignored for the residual structure.
  - c. For SOL 400, see Remarks [6](#) and [7](#).
5. The MPC option may be somewhat more expensive than the SPC option. However, it provides more realistic structural modeling than the SPC. When the MPC option is selected, the multipoint constraint may be applied on some degree of freedom for which the stiffness matrix is identified as singular. If the MPC is inapplicable to some degree of freedom, the SPC is used instead.
6. For SOL 400, if RESIDUAL option is requested, the AUTOSPC operation is applied to both the residual structure and the superelements. Without RESIDUAL option, the AUTOSPC operation is applied to the superelements only. For default (no AUTOSPCE command), AUTOSPC operation is not applied to the residual structure, but it is applied to the superelements. Both parameters PARAM,AUTOSPC and PARAM,AUTOSPCR have no effect in SOL 400. Please note that the AUTOSPC (RESIDUAL) command should not be used in the geometrical nonlinear analysis, because it may over constrain the structural model.
7. For SOL 400, the AUTOSPC (RESIDUAL) command can be placed above the subcase level, between subcase and step, and below the step level. The AUTOSPC operation is performed each step of a subcase if it is required. In the following example, step 10 uses SPC option, step 20 uses MPC option, and no AUTOSPC operation is performed for step 30.

```

SUBCASE 1
  STEP 10
    AUTOSPC (RESIDUAL, SPC) = YES
    LOAD = 10
  
```

```
STEP 20
  AUTOSPC (RESIDUAL,MPC) = YES
    LOAD = 20
STEP 30
  LOAD = 30
```

For superelements, only one AUTOSPC command can be specified. If there are multiple AUTOSPC commands in the Case Control packet, the one for the first step of the first subcase will be used. In the previous example, the AUTOSPC under step 10 is used.

8. For SOL 400, AUTOSPC equals YES for linear perturbation if DOMAINsolver ACMS is defined in the executive system.
9. For SOL 101 with contact, user can turn on AUTOSPC by using RESIDUAL option. i.e., when AUTOSPC(RESIDUAL)=YES, AUTOSPC will be turned on for linear analysis with contact.
10. This entry is not supported by SOL 600. Please enter PARAM,AUTOSPC in the bulk data.

**AUXCASE****Auxiliary Model Case Control Delimiter**

Indicates (delimits) the beginning of Case Control commands for an auxiliary model in SOL 200.

**Format:**

AUXCASE

**Examples:**

AUXCAS

AUXC

**Remarks:**

1. AUXCASE commands must follow the primary model Case Control commands.
2. All Case Control commands following this entry are applicable until the next AUXCASE command, or the BEGIN BULK delimiter. Commands from preceding Case Control Sections are ignored.
3. Each auxiliary model Case Control must be delimited with the AUXCASE command.
4. The AUXMODEL command is used to associate the auxiliary model Case Control with a particular auxiliary model.

## AUXMODEL

### Auxiliary Model Identification Number

References an auxiliary model for generation of boundary shapes in shape optimization.

**Format:**

AUXMODEL=n

**Examples:**

AUXMODEL=4

AUXM=4

| Descriptor | Meaning  |
|------------|--|
| n          | Auxiliary model identification number. (Integer > 0) |

**Remarks:**

1. AUXMODEL references a particular auxiliary model for analysis and may only be specified in the auxiliary model Case Control Section.
2. See the BEGIN BULK delimiter for the Bulk Data definition of an auxiliary model.

## AVLEXB

### Control for MSC Nastran-AVL EXCITE™ Interface

Control for MSC Nastran-AVL EXCITE™ Interface: (1) EXB file export from SOL 103 and SOL 400, (2) EXCITE™ results import and data recovery in SOL 111, 112 and 400.

Format:

```
AVLEXB [EXBBODY = {YES, NO}],  
[MASINVAR = {FULL, FIRST, NONE}],  
[RECOVRYM = {YES, NO}],  
[EXBOSET = {U1-U5, ALL}],  
[OUTGSTRS = {YES, NO}],  
[OUTGSTRN = {YES, NO}],  
[V1ORTHO = {-1.0, value1}],  
[V2ORTHO = {1.0e8, value2}],  
[MFFEXP = {YES, NO}],  
[NOD6 = {YES, NO}],  
[AVLPOST = {YES, NO}],  
[POSTUNT = {INP4_UNIT_NUM}],  
[EXBONLY = { YES, NO}]
```

Examples:

EXB file export

```
AVLEXB EXBBODY = YES
```

AVL EXCITE™ results import and data recovery

```
AVLEXB EXBBODY=NO AVLPOST=YES POSTUNT=130
```

| Descriptor | Meaning   |
|------------|---|
| EXBBODY    | Controls the output of MSC Nastran-AVL EXCITE™ (Default = NO): <ol style="list-style-type: none"> <li>1. NO: Do not output AVL EXB Flexible body.</li> <li>2. YES: Output AVL EXB Flexible body.</li> </ol>   |
| MASINVAR   | Requests the type of mass invariants to be computed (Default = FULL) : <ol style="list-style-type: none"> <li>1. FULL: All inertia invariants are computed.</li> <li>2. FIRST: Only first order inertia invariants are computed.</li> <li>3. NONE: No inertia invariants are computed.</li> </ol>   |
| RECOVRYM   | Controls the output of recovery (output transformation) matrix (Default = No) : <ol style="list-style-type: none"> <li>1. YES: Output A-set to (O+M)-set recovery matrix.</li> <li>2. NO: Do not output the A-set to (O+M)-set recovery matrix.</li> </ol>  |
| EXBOSET    | Selects a set of DOFs for controlling the output of recovery matrix (Default = ALL): <ol style="list-style-type: none"> <li>1. U1-U5: DOFs in the user defined set only are included in the O-set output to the EXB file and the partial recovery matrix output for these DOFs. These need to be defined in the bulk data using the USET/USET1 bulk data entry.</li> <li>2. ALL: Complete O-set in the model is retained for recovery.</li> </ol> |
| OUTGSTRS   | Outputs Grid Point Modal Stresses (Default = NO) to the MSC Nastran op2 requires PARAM,POST,1 or PARAM,POST,-1.   |
| OUTGSTRN   | Outputs Grid Point Modal Strains (Default = NO) to the MSC Nastran op2 requires PARAM,POST,1 or PARAM,POST,-1.  |
| V1ORTHO    | Lower frequency bound of the Craig-Bampton modes in cycles/unit time.<br><br>value1: Value of the lower bound.  |
| V2ORTHO    | Higher frequency bound of the Craig-Bampton modes in cycles/unit time.<br><br>value2: Value of higher bound.  |
| MFFEXP     | Requests output of Mass matrix of O+M set (Default = No). MFFEXP = YES requires the user to request full recovery (RECOVRYM=YES and EXBOSET = ALL).   |

| Descriptor | Meaning   |
|------------|---|
| NOD6       | If model is pure bar/beam elements with concentrated masses, this option can output the element dictionary table as well as element stiffness matrix (Default = No): <ol style="list-style-type: none"> <li>1. YES: Output KDICT &amp; KELM</li> <li>2. NO: Do not output KDICT &amp; KELM</li> </ol> If NOD6 = YES these parameters are automatically set MASINVAR = NONE, RECOVRYM = NO, MFFEXP = NO, EXBOSET = YES |
| AVLPOST    | Requests data recovery using the INP4 file generated by AVL EXCITE™ (Default = No): <ol style="list-style-type: none"> <li>1. YES: Request data recovery</li> <li>2. NO: data recovery</li> </ol> AVLPOST = YES requires EXBBODY = NO as the MSC Nastran AVL EXCITE™ interface cannot generate the EXB file and conduct data recovery in the same run.  |
| POSTUNT    | Unit number of the assigned INP4 file generated by AVL EXCITE™ which will be used for data recovery.  |
| EXBONLY    | Requests standard DMAP solution and data recovery following MSC Nastran-AVL EXCITE™ interface (Default = NO): <ol style="list-style-type: none"> <li>1. NO: Proceed with standard DMAP solution and data recovery after generating EXB file.</li> <li>2. YES: Executes only MSC Nastran-AVL EXCITE™ interface.</li> </ol>   |

**Remarks:**

1. It is highly recommended that the user should run MSC Nastran for Step-1 with maximum smp setting. Calculation of higher order mass invariants are very computation intensive; to make their calculation faster MSC Nastran leverages efficient shared memory parallelization (SMP).
2. To minimize the data storage and enable efficient data recovery in MSC Nastran, the use of MSC Nastran-AVL EXCITE™ Interface requires the use of EXTSEOUT feature. The use of this feature would be in its standard MSC Nastran capacity in SOL 103 or 400, e.g.:

```
ASSIGN OUTPUT2='crankextse.op2',UNIT=80,DELETE
...
EXTSEOUT (ASMBULK,EXTBULK,EXTID=20) DMIGOP2=80
...
```

During step-1 (EXB file export), to limit the size of external superelement (SE) .op2 the user should only request outputs for sets of physical quantities that are of interest. For, e.g., displacement and stress on surface nodes:

```
DISP(PLOT) = 101
...
STRESS(PLOT) = 102
```

above, the displacement and velocity can only be recovered on set 101 and stresses on set 102 during step-2 (data recovery).

DISP=ALL, STRESS=ALL, STRAIN=ALL, etc., should be used carefully for large models during step-1 as it could lead to significant performance degradation and large external SE .op2 file size.

3. AVL EXCITE™ supports the following flexbody types which will be set automatically based on the user inputs of AVLEXB case controls:
  - a. CON6: Is General large motion flexbody which is selected when the user enters “NOD6=NO” and “MASINVAR=FULL/FIRST”.
  - b. SMOT: Is Small motion flexbody which is selected when the user enters “NOD6=NO” and “MASINVAR=NONE”.
  - c. NOD6: Is a special flexbody model consisting of pure bar/beam (1-D) elements with concentrated masses. It is selected when the user enters “NOD6=YES”. Here the dictionary table for bar/beam elements for each bear/beam element shall be outputted into the EXB file. NOD6 automatically means that the following are not calculated and output into the EXB file: (1) Mass invariants, (2) Recovery Matrix, (3) A-set Orthonormalization and associated Eigenvalue/Eigenvector and (4) Component modes.
4. MSC Nastran does not export KELM to EXB file for NOD6 body. AVL EXCITE™ v2019 provides a utility to calculate and write KELM to existing EXB file.
5. Degree of freedom (dof) table (DOFTtable) is written to EXB file for the following dof sets:
  - a. A-set : Dof table for a-set are always written out to the EXB file.
  - b. O+M-set : Dof table for o+m-set are only written out to the EXB file if the recovery matrix is requested by the user. In case the user selects partial recovery by specifying the EXBOSET only a subset of o+m-set would be output in the dof Table.
6. GEOM table contains the coordinates of nodes in MSC Nastran basic coordinate system. Two table of coordinates are always written to the EXB file: (1) A-set and (2) F+M-set.
7. OUTGSTRS or OUTGSTRN entries require the use of the standard MSC Nastran STRESS= or STRAIN= Case Control commands used in conjunction with GPSTRESS= or GPSTRAIN= Case Control commands to produce modal grid point stress or strain. They also require the use of PARAM,POST,-1 or PARAM,POST,1. GSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain output to the MSC Nastran .f06 file. The modal grid point stress or strain are not output to the EXB file but to standard MSC Nastran .op2 file.
8. The default value of “MASINVAR = FULL” calculates all the mass invariants. In case of large models the calculations of higher order mass invariants can be time consuming; hence, it is recommended that the user first run with either “MASINVAR = FIRST” or “MASINVAR =NONE” to validate the model.
9. MFFEXP = YES requests output of Mass matrix of F+M set and requires the user to request full recovery (RECOVRYM=YES and EXBOSET = ALL).

10. Typically SOL 400 is used to produce a preloaded AVL EXCITE™ EXB file.
  - a. Standard SOL 400 nonlinear analysis prohibits the existence of an o-set; hence the EXTSEOUT case control must be used (see above Remark 2.).
  - b. In the preload run the structure should be statically supported and follower loading must be applied as a self equilibrating load set (not with SPC relationships!). In the ANALYSIS=MODES step, usually, the structure should be a free-free structure to preserve its six rigid body modes. In order to produce modal amplitudes and mode shapes and to ensure residual vector calculations, SPOINTs and Q-sets are required.

The example below is a typical SOL400 problem setup:

```
ASSIGN OUTPUT2='seplt3rbe.out2',UNIT=80,DELETE
SOL 400
CEND
$
$ Initiate an Nastran-AVL-EXCITE™ interface run
$ AVL REQUIRED ABOVE SUBCASE
AVLEXB EXBBODY=YES, EXBONLY=YES
SUBCASE 1
$ Preload
STEP 10
$ Static load and support for preload
SUBTITLE = PRELOAD
ANALYSIS = NLSTATICS
NLSTEP = 110
LOAD = 120
SPC = 130
BCONTACT = 140
SPCF = ALL
$ Generate stress and strain grid shapes
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$ Modal Step for Producing EXB file
$ Default: Select the end of previous load step to output
$ AVL EXCITE™ EXB
STEP 20
$ EXTSEOUT must be used to as Standard SOL 400 nonlinear analysis
$ prohibits the existence of an o-set
EXTSEOUT(ASMBULK,EXTBULK,EXTID=100) DMIGOP2=80
ANALYSIS = MODES
$ Select real Eigen Value Parameters
METHOD = 210
$ Turn residual vectors on
RESVEC = COMPONENT
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
```

- c. In the above example, the SPC set in the ANALYSIS=NLSTAT must be a static (nonredundant) constraint condition. Note that in the ANALYSIS=MODES STEP, the SPC constraint set has been removed. The definition of the attachment a-set for identifying attachment points and for q-set for specifying the desired number of modal amplitudes for orthonormalization should be done in the main bulk data section, e.g.:

```
$ Main Bulk section
$ Attachment point and component mode (A-SET) selection
ASET1,123456,1,11,111,121
QSET1,0,100001,THRU,100020
```

- d. If CONTACT is required as part of the preloading, it is highly recommended that the friction option be turned on by using an appropriate BCPARA bulk data entry setting, e.g.,

```
$ Select bilinear Coulomb friction for all subcases
BCPARA, 0, FTTYPE, 6
```

If contact friction is not turned on, the tangential motion between the two parts coming into contact will most likely not be constrained and incorrect or fatal results will occur.

11. The AVL EXCITE™ solver expects singularities, other than the rigid body modes, of the model to be constrained; hence, it is recommended that the user constrain singularities automatically using the PARAM,AUTOSPC in SOL 103 and AUTOSPC (RESIDUAL) command in SOL 400.
12. After conducting the AVL EXCITE™ simulation the data recovery is conducted in SOL 111, SOL 112 and SOL 400 using the EXTSEOUT feature. Using External SE to conduct data recovery would make the data recovery faster and minimize the database size by storing only the information required for recovery. The user interface for data recovery is straightforward requiring the attachment of the SE databases, stored on the .op2 file, generated in the SE generation step during EXB file export together with the ASM and PCH files.
13. During the data recovery it should be noted that PARAM,POST setting for the External SE are not picked up from the main bulk. If physical quantities inside External SE (stresses,etc) are to be output to op2 the user can add PARAM,POST in the case control if they want it applied globally or add PARAM,POST manually to the External SE pch file.
14. Virtual mass is supported in MSC Nastran-AVL EXCITE™ Interface through the use of standard VMOPT parameter entry.
15. Real coupled method does not support in AVL EXCITE™ interface. If Case Control METHOD(COUPLED) is used with AVLEXB EXBBODY=yes, the following error message will be returned:  
\*USER FATAL MESSAGE 22984 (SUBDMAP IFPL)  
CASE CONTROL COMMAND METHOD(COUPLED) IS NOT SUPPORTED WITH THE NASTRAN/AVL EXCITE INTERFACE.

**AXISYMMETRIC****Conical Shell Boundary Conditions**

Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.

Format:

$$AXISYMMETRIC = \left\{ \begin{array}{l} SINE \\ COSINE \\ FLUID \end{array} \right\}$$

**Example:**

AXISYMMETRIC=COSINE

| Descriptor | Meaning                                  |
|------------|--|
| SINE       | Sine boundary conditions will be used.   |
| COSINE     | Cosine boundary conditions will be used. |
| FLUID      | Existence of fluid harmonics.            |

**Remarks:**

1. This command is required for conical shell problems.
2. If this command is used for hydroelastic problems, at least one harmonic must be specified on the AXIF command.
3. See the [Surface Elements](#) in the *MSC Nastran Reference Guide* for a discussion of the conical shell problem.
4. The sine boundary condition will constrain components 1, 3, and 5 at every ring for the zero harmonic.
5. The cosine boundary condition will constrain components 2, 4, and 6 at every ring for the zero harmonic.
6. SPC and MPC Case Control commands may also be used to specify additional constraints. See [Case Control Commands, 191](#).

## BC              Boundary Condition Identification

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

**Format:**

BC=n

**Example:**

BC=23

| Descriptor | Meaning                              |
|------------|--------------------------------------|
| n          | Identification number (Integer > 0). |

**Remarks:**

1. In SOLs 103, 105, 145, and 200, BC is required in each subcase if multiple boundary conditions are specified for normal modes, buckling, or flutter analysis.
2. If only one boundary condition is specified, then BC does not have to be specified, and n defaults to zero.

**B2GG**

## Direct Input Damping Matrix Selection

Selects direct input damping matrix or matrices.

**Format:**

B2GG=name

**Examples:**

B2GG = BDMIG

B2GG = BDMIG1, BDMIG2, BDMIG3

B2GG = 1.25\*BDMIG1, 1.0\*BDMIG2, 0.82\*BDMIG3

SET 100 = B1, B2

B2GG = 100

| Descriptor | Meaning   |
|------------|---|
| name       | Name of $[B_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list, with or without factors (see Remark 5.). |

**Remarks:**

1. DMIG matrices will not be used unless selected.
2. Terms are added to the damping matrix before any constraints are applied.
3. The matrix must be symmetric, and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
4. A scale factor may be applied to this input via the PARAM, CB2 entry. See [Parameters, 783](#).
5. The formats of the name list:

- a. Names without factor.

Names separated by comma or blank.

- b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be with a factor including 1.0.

**B2PP****Direct Input Damping Matrix Selection**

Selects direct input damping matrix or matrices.

**Format:**

B2PP=name

**Example:**

B2PP = BDMIG

B2PP = BDMIG1, BDMIG2, BDMIG3

B2PP = 5.06\*BDMIG1, 1.0\*BDMIG2, 0.85\*BDMIG3

B2PP = (1.25, 0.5) \*BDMIG1, (1.0, 0.0) \*BDMIG2, (0.82,-2.2) \*BDMIG3

| Descriptor | Meaning   |
|------------|---|
| name       | Name of $[B_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list, with or without factors. See Remark 7. (Character). |

**Remarks:**

1. DMIG entries will not be used unless selected.
2. B2PP is used only in dynamics problems.
3. DMIAX entries will not be used unless selected by the B2PP command.
4. The matrix must be square or symmetric, and field 4 on the DMIG,name Bulk Data entry must contain a 1 or 6.
5. It is recommended that PARAM,AUTOSPC,NO be specified. See the [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
6. The matrices are additive if multiple matrices are referenced on the B2PP command.
7. The formats of the name list:
  - a. Names without factor  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parentheses as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

**BC**      Boundary Condition Identification

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

Format:

BC=n

Example:

BC=23

| Descriptor | Meaning                              |
|------------|--------------------------------------|
| n          | Identification number (Integer > 0). |

Remarks:

1. In SOLs 103, 105, 145, and 200, BC is required in each subcase if multiple boundary conditions are specified for normal modes, buckling, or flutter analysis.
2. If only one boundary condition is specified, then BC does not have to be specified, and n defaults to zero.

## BCHANGE

## Contact Bodies Definition Change Selection

Selects the changes of the definition of contact bodies in SOL 400.

**Format:**

BCHANGE=n

**Example:**

BCHANGE=10

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification of the BCHANGE Bulk Data entry, see Remark 2. (Integer > 0) |

**Remarks:**

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of the BCHANGE Bulk Data entry is defined on the BCONTACT Case Control command if applicable; however, the SID on the BCHANGE Case Control command can overwrite it.

**BCONCHK****Contact Model Check**

This entry is used to activate contact model check before analysis in SOL 101, 103, 105, 107~112, 200 and 400.

With the contact model check, the initial contact status is checked and output. Displacement-like vectors and norm distance of active nodes to the corresponding contact body and distance to the body are generated. Sign of the distance stands for its status: positive for gap; 0.0 on surface; negative for penetration. It is a global case control and must be above SUBCASE.

Note that BCONCHK does not support BCONTACT=ALLBODY, i.e., no contact status check is reported when BCONTACT=ALLBODY.

Format:

$$\text{BCONCHK} \begin{bmatrix} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{bmatrix} = \begin{Bmatrix} \text{RUN} \\ \text{STOP} \\ \text{STEP} \end{Bmatrix}$$

**Example:**

BCONCHK (PRINT) = Run

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|                  | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |      |  |
|------|--|
| RUN  | Execute contact model check and proceed analysis normally.   |
| STOP | Only run contact check and exit job before normal analysis.  |
| STEP | Execute contact check at each output request LOAD/TIME step. |

**BCONTACT**

Selects 3D Contact Surfaces

This entry activates contact analysis and selects contact definition tables.

#### Overview of contact type definition:

**General Contact** - two bodies may come into contact and separate at any point in the simulation and the bodies may slide along each other's surface with or without friction. The fundamental constraint is no relative normal displacement when bodies are in contact. General contact is available in SOL 101, SOL 400, SOL 600 or SOL 700 only. For SOL 101, contact is the only source of nonlinearity, no large displacements, large rotations or material nonlinearity is available.

**Glued Contact** - two bodies may come into contact and separate at any point in the simulation, but when in contact there is no relative sliding. The fundamental constraint is no relative normal displacement or tangential displacement when bodies are in contact. One can consider this equivalent to two surfaces that have infinite friction. Note that the word glued, only refers to the constraint on the tangential behavior. Bodies that are in glued contact may lose contact if the separation (force or stress based) separation criteria is exceeded, due to Unglue or Breaking Glue.

**Step Glued Contact** - this is available for SOL 400 only. Step Glued contact is activated using a value a negative value of IGLUE for each contact pair. It is similar to Glued Contact, there are two conditions.

1. The contact status will be checked at the beginning of the step, and those nodes or segments that are in contact will remain in glued contact for the entire step. The constraints will change due to large rotations. Furthermore if a large tensile force or stress developed over the interface in the current loadcase, no separation would occur for these regions which are initially in contact. . Performing an Unglue or Breaking glue would also not be enforced during the step for these regions. This may be successfully used to model the union of dissimilar meshes, where at a later time one wanted to separate the bodies (like opening of a door).
2. When using Step Glue conventional contact occurs for the nodes/segments of the body which are not in contact at the beginning of the step. That means when they come into contact, they will glue, but they may separate within the same step.

**Permanently Glued Contact** -this is a special case of contact, where the initial configuration is used to determine the contact constraint, and these contact changes due not change throughout the analysis. Nodes or segments which are not initially in contact do not come into contact, and in fact may penetrate the model. The constraint is a glue type, meaning there will be no relative normal or tangential displacement. Permanent glued may be used to connect dissimilar meshes or for simple assembly modeling when no other contact occurs. The bodies will never separate. Permanent glue should not be used in models that experience large rotations. It is applicable to SOL 101, 103, 105, 107, 108, 109, 110, 111, 112, SOL 200 and SOL 400. Permanent Glued contact is activated if the BCTABLE or BCTABL1 that is referenced in the first Loadcase (SOL 100\*) or in the first Step (SOL 200) has a value of IGLUE greater than zero for all contact body pairs.

If the user requires conventional (general) contact for the complete simulation, but Permanent Glued contact is invoked, enter bulk data BCPARA,0,NLGLUE,1 to deactivate the Permanent Glue in a subsequent step.

If large rotation/deformation effect present, turn on SYSTEM(758)=2 will switch Permanently Glued Contact to Step Glued Contact automatically in SOL 400.

Because glued contact is very useful in assembly modeling problems encountered in engineering practice, several special cases are considered as well.

**Moment Carrying Glue** - For General Contact - when a node or surface comes into contact the default condition is that constraints are only placed on the translational degrees of freedom. When Moment Carrying Glue is activated the rotational degrees of freedom are also constrained to ensure moment carrying behavior. This may be used with either beam elements or shell elements.

**Symmetry Contact** - Deformable contact with a rigid Symmetry surface. In this case no friction is allowed. Furthermore no separation is allowed, and finally if the rigid surface is contacted by beams or shells - the rotations are automatically constrained to satisfy the symmetry constraints. You do not need to specify any additional input, other than specifying that the rigid surface is a SYMM body.

Format:

$$BCONTACT = \begin{cases} n \\ ALLBODY \\ NONE \end{cases} \text{ or } \begin{cases} ALLELE \end{cases}$$

↑  
SOL 700 only

Examples:

```
BCONTACT = 5
BCONTACT=ALLBODY
BCONTACT=AUTO, PGLUE
```

| Descriptor | Meaning   |
|------------|---|
| n          | Identification number of a BCTABLE, BCTABL1, BCONECT, BCHANGE, and/or BCMOVE Bulk Data entry.   |
| ALLBODY    | All bodies defined using all the BCBODY entries can potentially contact each other. This option can only be used if it applies to all subcases. (all SOLs) In SOL 400, when BCONTACT=ALLBODY, in any STEP, all BCBODY entries defined in the model can potentially contact each other. In this case, no BCTABLE, BCTABL1, BCONECT is required. See also Remark 3. |

| Descriptor  | Meaning   |
|-------------|---|
| NONE        | All contact definitions (BCTABLE, BCTABL1, BCNECT, BCBODY, and BCBODY1) are ignored. For SOL 400 and SOL 101, if BCONTACT = NONE is entered in any subcase, it applies for all subcases. (Default for SOLs 101 and 400.)  |
| AUTO(ctype) | <p>Automatic generate contact. ctype is optional. It may be</p> <p>TOUCH (default) -- general touching contact;</p> <p>PGLUE -- Permanently glued contact;</p> <p>GGLUE -- General Glued Contact; and</p> <p>SGLUE -- Step Glued Contact.</p> <p>It works with bulk data section entry BCAUTOP, BCAUTOP is optional. See remark <a href="#">5</a>. and <a href="#">6</a>.</p> |

**Remarks:**

1. BCONTACT is recognized in SOLs 101, 400, and 700, and under the special condition of permanent glued contact in SOLs 103, 105, 107, 108, 109, 110, 111, 112 and 200. The standard SOL 200(without calling SOL 400) can only support permanent glue; but when SOL 200 calls SOL 400(or say SOL 400 optimization), it can support all contact types.
  2. For SOLs 101 and 400, if the form BCONTACT=n is applied in any loadcase (subcase or step), Nastran looks into the Bulk Data file to get all BCTABLE, BCTABL1 or BCNECT (required), BCMOVE (optional) and BCHANGE (optional), in the same SID=n. The user can always specify Case Control commands, BCMOVE and/or BCHANGE, to select different SID.
- BCONTACT=0 can invoke initial preload contact conditions, such that the contact bodies will just touch each other before analysis begins, but it's not necessary. If presence of BCTABLE, 0, initial contact condition will also be processed, no matter if BCONTACT=0 is given or not. (please refer to BCTABLE, 0 for more information).

The contact analysis during any loadcase is dominated by BCONTACT=n (n>0). Without presence of BCONTACT=n (n>0), no contact analysis works in that particular loadcase.

In SOL 400 one can have multiple contact interaction types in the model meaning general contact, glued contact, step contact on a contact pair basis. These can change from step to step by activating a new BCTABLE or BCTABL1.

SOL 700 allows only one BCONTACT Case Control command and only one subcase.

3. For SOLs 101 and 400, if the form BCONTACT=ALLBODY is applied in any loadcase (subcase or step), Nastran does not look into corresponding BCTABLE, BCTABL1 or BCNECT but uses the defaults for all entries on BCTABLE, BCONPRG or BCONPRP. If the user wants to specify BCMOVE and/or BCHANGE Bulk Data entries, the BCMOVE and/or BCHANGE Case Control commands must be given.

4. Permanent glued contact for small deformation and rotation is initiated by BCNCONTACT=n pointing to BCTABLE/BCONNECT referring a valid positive IGLUE field.  
If all slave's for the BCTABLE or BCONPRG corresponding to the first loadcase (first subcase and first step) contain IGLUE >0, permanent glued contact with small rotation condition will be used for all SLAVE entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified.  
If IGLUE < 0 exists, step glue is activated to support large deformation and large rotation.
5. For glue contact (PGLUE, GGLUE, and SGLUE), default is to implement the moment carrying glue without node projection, i.e., Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. Unless IGLUE is defined in BCAUTOP.
6. CTYYPE can be specified by bulk data section BCAUTOP also, the value of CTYYPE in BCAUTOP will be used if it is specified.
7. When BCNCONTACT=AUTO is specified, acg file will be generated. An acg file is a file consists of all generated contact bodies, contact pairs and parameters. The name of acg file can be specified by ASSIGN FMS statement.
8. When BCNCONTACT=AUTO is specified, only elements in residual structure are used to construct contact bodies in superelement models.
9. BEAMB in bulk data entry BCPARA is set to 1 when BCNCONTACT=AUTO is specified.

**BCONTACT**

Selects 3D Contact Surfaces in SOL 600

This entry is used to initiate and control 3D contact in SOL 600.

Standard subcase rules apply and BCONTACT may be specified within each subcase to define which bodies may make contact during that particular subcase. A BCONTACT = 0 above the subcase level is used in SOL 600 to invoke an option such that the contact surfaces will just touch each other before the nonlinear simulation begins. It is highly recommended that if contact is specified for any subcase in SOL 600, and if there are any rigid contact surfaces, BCONTACT = 0 and a matching BCTABLE with ID = 0 be included.

Format:

$$BCONTACT = \begin{cases} n \\ ALLBODY \\ NONE \end{cases} \text{ or } \begin{cases} ALL \\ ALLELE \\ ALLELE2 \\ ALLGLUE \\ ALLGLU2 \\ ALLGLUM \\ ALLGLM4 \\ ALLGLUP \\ BCBOX \\ BCPROP \\ BCMATL \end{cases}$$

Examples:

BCONTACT = 5  
BCONTACT=ALLBODY

| Descriptor | Meaning  |
|------------|--|
| n          | Identification number of a BCTABLE, BCHANGE, and/or BCMOVE Bulk Data entry. If the model has beams, this option or BCPROP or MCMATL must be used.  |
| ALLBODY    | All bodies defined using all the BCBODY entries can potentially contact each other. This option can only be used if it applies to all subcases.  |
| NONE       | All contact definitions (BCTABLE, BCBODY) are ignored. BCONTACT = NONE may be used for any subcase desired and/or for increment zero - some subcases can have contact and others no contact.   |
| ALL        | All elements in the model can potentially contact with each other (Default). When this option is specified, no 3D contact input is required in the bulk data and, if entered, will be ignored. Warning—this option may take excessive computer time. This option can only be used if it applies to all subcases. |
| ALLELE     | Same as ALL. All elements in the model must be the same kind (either shell or solid). This option may not be used if there are beams, bars or rods in the model.   |

| Descriptor | Meaning  |
|------------|--|
| ALLELE2    | Same as ALLELE except shell and solid elements may both be in the model in which case separate bodies for each are formed. (This option requires that both shells and solids be in the model.) All beams, bars and rods in the model will be ignored with respect to contact.  |
| ALLGLUE    | Same as ALLELE or ALLELE2 except glued contact will be used for all bodies. The glued contact can vary from increment to increment, however when grids come into contact they cannot separate. BCBODY and BCTABLE entries are created automatically. BCBODY slave entries have IGLUE=1 (see BCTABLE entry).  |
| ALLGLU2    | Same as ALLELE or ALLELE2 except glued contact will be used for all bodies. The glued contact can vary from increment to increment, however when grids come into contact they cannot separate. BCBODY and BCTABLE entries are created automatically. BCBODY slave entries have IGLUE=2 (see BCTABLE entry).  |
| ALLGLUM    | Same as ALLELE or ALLELE2 except glued contact with moment carrying glue will be used for all bodies. BCBODY and BCTABLE entries are created automatically. BCBODY slave entries have IGLUE=3 (see BCTABLE entry).   |
| ALLGLM4    | Same as ALLELE or ALLELE2 except glued contact with moment carrying glue will be used for all bodies. BCBODY and BCTABLE entries are created automatically. BCBODY slave entries have IGLUE=4 (see BCTABLE entry).   |
| ALLGLUP    | Same as ALLELE or ALLELE2 except “permanent glue” is used. This option determines which grids are initially in contact and uses this contact situation in a glued condition for the remainder of the analysis. To use this option, specify PERMGLUE on the SOL 600 Executive Control statement. (SOL 600 only) Note for SOL 600, the PERMGLUE option is the only way contact can be used with SOL 600,101 or SOL 600,103 or other “linear” analyses. |
| BCBOX      | All elements defined within a box-like region as defined by the Bulk Data entry BCBOX can potentially contact each other. See Remark 2.  |
| BCPROP     | All elements defined by the Bulk Data entry BCPROP can potentially contact each other. See Remark 2.   |
| BCMATL     | All elements defined by the Bulk Data entry BCMATL can potentially contact each other. See Remark 2.   |

### Remarks:

1. Normally, only one form of this entry may be used in any given analysis. Analysis restarts must use the same form as the original run. An exception is that if BCONTACT = NONE is entered for any subcase, BCONTACT=N may also be specified for different subcases. BCONTACT=ALLxxx cannot be mixed with BCONTACT=NONE or BCONTACT=N in the same input file.
2. Bulk Data entries BCBOX, BCPROP, and BCMATL may be used with BCONTACT = n, wherein case IDs specified on the BCBODY entry and on the BCBOX, BCPROP, and/or BCMATL entries must match.

3. For options ALLELE, ALLELE2, ALLGLUE, ALLGLU2, ALLGLUM, and ALLGLM2, no BCBODY or BCTABLE entries should be included in the model. Nastran will automatically create BCBODY and/or BCTABLE entries as necessary for these options.
4. The ALLELE option requires that the model be made up of either all shells or all solids.
5. The ALLELE2 option requires that the model have both shells and solids.
6. The ALLGLUE, ALLGLU2, ALLGLUM and ALLGLM4 options can be used with models with shells only, solids only for a combination of shells and solids.
7. The ALLGLUP option can either be used with BCBODY/BCTABLE entries or without them. If there are not BCBODY/BCTABLE entries, SOL 600 will create them. If they exist, all SLAVE lines on all BCTABLE entries must specify IGLUE=1.
8. The ALLG\* options are not available for edge contact. Edge contact may only be run using BCONTACT=N and the corresponding BCTABLE having a FBSH line with COPS1 and/or COPM1 properly specified.

**BCMOVE**

## Contact Body Movement Selection

Selects movement of bodies in contact in SOL 400.

**Format:**

BCMOVE=n

**Example:**

BCMOVE=10

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of the BCMOVE Bulk Data entry. (Integer > 0) |

**Remarks:**

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of the BCMOVE Bulk Data entry is defined on the BCONTACT Case Control command if applicable; however, the SID on the BCHANGE Case Control command can overwrite it.

**BEGIN BULK****Case Control and Bulk Data Delimiter**

Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.

Format:

|                    |   |
|--------------------|---|
| <b>BEGIN[BULK]</b> | <i>AFPM = afpmid</i><br><i>ARBMODEL = arbmid</i><br><i>AUXMODEL = auxmid</i><br><i>MASSID = massid[LABEL = masslabel]</i><br><i>MODULE= moduleid[APPEND][LABEL = modlabel]</i><br><i>FLXBDY = flexbody</i><br><i>SUPER = seid</i><br><i>TRMC = trimid</i><br><i>UDS</i> |
|--------------------|---|

Examples:

```
BEGIN BULK
BEGIN AUXMODEL=22
```

```
BEGIN BULK TRMC=101
BEGIN TRMC=102
```

| Descriptor | Meaning  |
|------------|--|
| AFPM       | Indicates the beginning of an acoustic field point mesh Bulk Data Section.   |
| afpmid     | Acoustic field point mesh identification number (Integer > 0).   |
| ARBMODEL   | Indicates the beginning of a finite element model FEM for an arbitrary beam cross-section.   |
| arbmid     | FEM identification number (Integer > 0).   |
| AUXMODEL   | Indicates the beginning of an auxiliary model Bulk Data Section.   |
| auxmid     | Auxiliary model identification number (Integer > 0).   |
| FLXBDY     | Indicates the beginning of a Adams/Nastran MNF flexbody Bulk Data Section for SOL400 analysis.   |
| flexbody   | Flexbody MNF component identification number (Integer>0; Required). Component usually consists of ASET and QSET description for attachment points. |
| MASSID     | Indicates the beginning of the incremental mass case Bulk Data Section.  |
| massid     | Incremental mass case identification number (Integer>0; Required).   |
| LABEL      | Indicates a massid label.  |
| MODULE     | Indicates the beginning of a Module Bulk Data section  |
| moduleid   | Module identification number (Integer>0). See Remark 11.   |

| Descriptor | Meaning  |
|------------|--|
| APPEND     | Indicates append data to a module.   |
| LABEL      | Indicates a Module label. This option will be converted to an MDLABEL Bulk Data entry in the main Bulk Data section. |
| modlabel   | Module label.  |
| SUPER      | Indicates the beginning of partitioned superelement Bulk Data Section.   |
| seid       | Superelement identification number (Integer $\geq 0$ ).  |
| TRMC       | Indicates the beginning of Bulk Data Section for a trim component model. See Remark 9.                               |
| trimid     | Trim component identification number (Integer $> 0$ ; Required).   |
| UDS        | Indicates the beginning of user defined subroutine section.  |

#### Remarks:

1. BEGIN BULK is not required but it is highly recommended that the user supply it. If not specified, then the program will automatically insert one, before the first unique bulk type entry. Parameters are not considered unique bulk type entries so automatic BEGIN BULK inclusion will come after any parameter that precedes entries unique to bulk.
2. For an auxiliary model, AUXMID is referenced by the AUXMODEL Case Control command.
3. Partitioned Bulk Data Sections defined by BEGIN SUPER are used to define only one superelement each. Bulk Data commands which define superelements are ignored in partitioned Bulk Data Sections.  
Superelements specified by a BEGIN SUPER entry can be automatically attached to other superelements based on relative locations of grid points. For connection to the downstream superelement, the global coordinate directions of the attachment grid points of the upstream superelement will be internally transformed to the global coordinate directions of the grid points of the downstream superelement. For displacement data recovery, the output will be in the original global coordinate directions.
4. An acoustic field point mesh section defined by BEGIN AFPM is used to define one acoustic field point mesh. Acoustic field point meshes are used for postprocessing of acoustic results in the far field only; i.e., at locations within the acoustic infinite elements.
5. The BEGIN SUPER, BEGIN AUXMODEL, and BEGIN AFPM Bulk Data entries must lie between BEGIN BULK and ENDDATA.
6. When employing part superelements using the BEGIN BULK SUPER (or BEGIN SUPER) entry, it should be noted that any parameters that are specified in the main Bulk Data Section apply only to the residual and not to any of the part superelements. Accordingly, to apply certain parameters to all of the superelements, they must be specified in the Case Control Section, or explicitly in all of the BEGIN BULK SUPER (or BEGIN SUPER) portions of the Bulk Data. A common example of such a parameter specification is PARAM,POST, which is used to request postprocessing of results.

7. arbmid can be referenced under the OUTM keyword of the PBMSECT Bulk Data entry to define the geometry of the arbitrary beam cross-section.
8. For model using user defined subroutines, the subroutines can be put in the UDS section. Nastran can build user service from the subroutines using command line keyword uds=model. See the document User Defined Service User's Guide for detail.
9. The parameter WTMASS is supported for trim components with a PARAM, WTMASS, value in each BEGIN TRMC=n Bulk Data.
10. BEGIN MODULE command can only be specified after the BEGIN BULK or an ENDMODULE command has already been specified. In other words, a Module Bulk Data section cannot be nested inside another Module's Bulk Data section.
11. The same module identification number can appear multiple times if APPEND is specified, otherwise a user fatal message will be issued.

**BENDL**

## BAR/BEAM end load Subcase Selection

Function to calculate bar end loads and shear flow.

Format:

$$BENDL \left( PRINT = \begin{array}{l} YES \\ NO \end{array} \right) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

To output a table of bar end loads/shear flows.

BENDL = ALL

To output a table of bar end loads/shear flows and print table to F06.

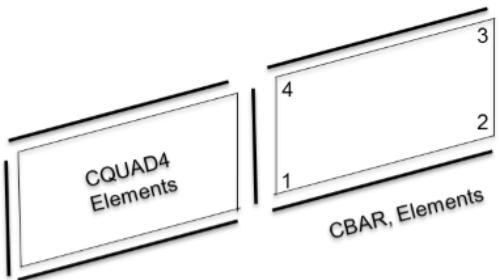
BENDL (PRINT) = ALL

| Descriptor | Meaning  |
|------------|--|
| PRINT      | Prints bar end loads and panel shear flow to the F06 file.   |
| YES        | BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will be printed.               |
| NO         | BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will not be printed. (Default) |
| ALL        | Select all bar/beam/rod type elements associated with “Shell” panels.  |
| n          | Identification of a previously defined SET listing “Flange/Boom” elements.                                   |
| NONE       | Turn off BENDL in selected subcases.   |

Remarks:

1. A BENDL Case Control Command is to appear in any subcase for which Bar End Loads and Panel shear flows are required. This entry is only applicable in SOL101 or ANALYSIS=STATIC.
2. The presence of BENDL in a linear statics run will automatically turn on GPFORCE=ALL as well as PARAM, NOELOF, 1.
3. Only CSHEAR, CQUAD4, CTRIA3, CQUADR, and CTRIAR panels will contribute to shear flow and bar end load calculations.
4. Include MDLPRM HDF5 to output OUTSFLOW data block to the HDF5 output file.

5. The following figure shows a typical bar end load and shear flow panel.



6. Print and table FORMAT is:

```
B A R   E N D   L O A D S   A P P R O X I M A T I O N S   ( C B A R )
ELEMENT      TOTAL END LOADS
ID.          END 1           END 2
  6       -1.994413E+03 -1.136868E-13
 11        2.082592E+03  3.638604E-01
 16       -4.721992E+02 -7.329887E+01
 21        4.984904E+02 -4.987337E+02

B A R   E N D   L O A D S   S H E A R   F L O W   A P P R O X I M A T I O N S ( C
Q U A D 4 )
ELEMENT ID.          SHEAR 12          SHEAR 23          SHEAR 34          SHEAR 41
          1       9.972063E+01 -2.493060E+02 -9.972063E+01  3.989003E+01
```

**BOUTPUT**

## Line Contact or 3D Contact Output Requests

Selects contact regions for output.

Format:

$$BOUTPUT \left[ \begin{pmatrix} SORT1 \\ SORT2 \end{pmatrix}, \begin{pmatrix} PRINT, PUNCH \\ PLOT \end{pmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

BOUTPUT=ALL

BOUTPUT=5

| Descriptor       | Meaning   |
|------------------|---|
| SORT1            | Output is presented as a tabular listing of slave nodes for each load or time depending on the solution sequence. |
| SORT2            | Output is presented as a tabular listing of load or time for each slave node.                                     |
| PRINT or (blank) | Printer File (.f06)   |
| PUNCH            | Punch File (.pch)   |
| PLOT             | Plot File (.op2/.h5)  |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |  |
|------|--|
| ALL  | Histories of all the slave nodes (all nodes for 3D Contact) listed in all the BOUTPUT Bulk Data entries are output. If no BOUTPUT Bulk Data entries are specified, histories of all the slave nodes in all the contact regions are output.   |
| n    | Set identification of previously appearing SET command. Only contact regions with identification numbers that appear on the SET command are selected for output. If there is a BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for slave nodes listed in the Bulk Data entry are output. If there is no BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for all the slave nodes in that contact region are output. |
| NONE | Result histories for slave nodes are not calculated or output.   |

**Remarks:**

1. BOUTPUT is processed in SOLs 101, 106, 129, 153, 159, 400, and 600 only.
2. SORT1 is the default in SOLs 106 and 153. SORT2 is the default in SOLs 129 and 159.
3. Only SORT1 is available for 3D Contact.
4. The initial contact status may be output when presence of BCTABLE/ID=0. Please note that all the contact force/stress in the initial state are zero. Only the contact status is relevant.

**BSQUEAL**

## Brake Squeal Analysis Data Selection

Selects data for brake squeal analysis in SOL 400.

**Format:**

BSQUEAL= n

**Example:**

BSQUEAL=10

| Describer | Meaning   |
|-----------|---|
| n         | Set identification number of a BSQUEAL Bulk Data entry (Integer > 0). |

**Remark:**

This command is used only in SOL 400 for brake squeal analysis with 3D Contact.

This command is not available with segment-to-segment contact.

## CAMPBELL

### Campbell Diagram Parameters

Specifies Campbell Diagram parameters.

**Format:**

CAMPBELL= n

**Example:**

CAMPBELL= 10

| Descriptor | Meaning   |
|------------|---|
| n          | Identification number of a CAMPBLL Bulk Data entry (Integer > 0). |

**Remark:**

1. CAMPBELL option is supported for both SOL 107 and SOL 110 analysis.
2. When there is a Case Control request for Campbell diagram, the selected RGYRO Bulk Data entry must have the ASYNC option specified in its SYNCFLG field (Field 3). Otherwise, the program terminates the execution with an appropriate fatal message.

**CLOAD****Static Load Request for Upstream Superelement Loads**

Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.

**Format:**

CLOAD=n

**Example:**

CLOAD=15

| Descriptor | Meaning  |
|------------|--|
| n          | Identification number of a unique CLOAD Bulk Data entry (Integer > 0). |

**Remarks:**

1. This command may only appear in the residual structure subcases (see the Case Control command, [SUPER \(Case\), 604](#)) and, if used, it must be specified in all of them.
2. The CLOAD Bulk Data entry must reference previously processed LSEQ (load sequence) Bulk Data that was requested by LOADSET Case Control commands on the upstream ( $SUPER \neq 0$ ) subcases.
3. The resulting load is added to those produced by LOAD and TEMP(LOAD) Case Control commands in the residual structure subcases.

## CMETHOD

## Complex Eigenvalue Extraction Method Selection

Selects complex eigenvalue extraction parameters.

**Format:**

CMETHOD=n

**Example:**

CMETHOD=77

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification of EIGC (and EIGP) Bulk Data entry (Integer > 0). |

**Remarks:**

1. The CMETHOD command must be specified in order to compute complex eigenvalues.
2. See description of the parameter, [UNSYMF, 1003](#), to perform complex eigenvalue analysis in SOL 106.

**CMSENRGY****Component Modal Synthesis Energy Output Request**

Requests the form and type of component modal synthesis (CMS) energy output.

Format:

$$\begin{aligned}
 CMSENRGY( & \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right], \left[ \begin{array}{c} REAL \text{ or } IMAG \\ PHASE \end{array} \right], \left[ \begin{array}{c} ESORT = \left\{ \begin{array}{c} MODE \\ ASCEND \\ RATIO \end{array} \right\} \end{array} \right], \\
 & \left[ \begin{array}{c} RESPONSE = \left\{ \begin{array}{c} BOTH \\ MODAL \\ FORCED \end{array} \right\} \end{array} \right], \left[ \begin{array}{c} CMSE = \left\{ \begin{array}{c} ALL \\ NONE \\ TOTAL \\ QSET \end{array} \right\} \end{array} \right], \left[ \begin{array}{c} CMKE = \left\{ \begin{array}{c} ALL \\ NONE \\ TOTAL \\ QSET \end{array} \right\} \end{array} \right], \\
 & \left[ \begin{array}{c} CMDE = \left\{ \begin{array}{c} ALL \\ NONE \\ TOTAL \\ QSET \end{array} \right\} \end{array} \right], \left[ \begin{array}{c} FILTER = \left\{ \begin{array}{c} 0.001 \\ fratio \end{array} \right\} \end{array} \right], [TOPN = m] ) = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

**Example:**

```
CMSENRGY (PHASE, RESPONSE=FORCED, CMSE=TOTAL, CMKE=QSET) = ALL
SET 1001 = 10,40
CMSENRGY (PUNCH, PRINT, RESPONSE=BOTH, CMSE=ALL, FILTER=0.01) = 1001
```

| Descriptor       | Meaning             |
|------------------|---------------------|
| PRINT or (blank) | Printer File (.f06) |
| PUNCH            | X                   |
| PLOT             |                     |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|        |  |
|--------|--|
| ESORT  | Keyword selecting one of the output sorting options: MODE, ASCEND, or RATIO. |
| MODE   | Results are output in order of increasing CMS natural mode number.           |
| ASCEND | Results are output in order of increasing energy ratio magnitudes.           |
| RATIO  | Results are output in order of decreasing energy ratio magnitudes.           |

| Descriptor | Meaning   |
|------------|---|
| RESPONSE   | Keyword selecting the types of results to be produced.  |
| BOTH       | Requests output for both free and forced response solutions.  |
| MODAL      | Specifies that output is to be generated for the free (or real eigenvalue) response solution.                   |
| FORCED     | Specifies that output is to be generated for the forced response (modal frequency or modal transient) solution. |
| CMSE       | Keyword requesting output of CMS strain energy ratios.  |
| CMKE       | Keyword requesting output of CMS kinetic energy ratios.   |
| CMDE       | Keyword requesting output of CMS damping energy ratios.   |
| ALL        | Requests both TOTAL and QSET output.  |
| NONE       | Requests that no CMS energy output be generated.  |
| TOTAL      | Requests CMS energy ratio totals in all component modes of a superelement.                                      |
| QSET       | Requests CMS energy ratios for individual component modes.  |
| FILTER     | Keyword specifying the value of the printed output data filter.   |
| fratio     | Value of output filter ratio (Default = 0.001).   |
| TOPN       | Keyword specifying the number of largest CMS energy ratios to be output.  |
| m          | The number of largest CMS energy ratios to be output (Default is all ratios).                                   |
| n          | Results for superelement IDs in SET n will be output.   |
| ALL        | Results for all recovered superelements will be output.   |
| NONE       | No CMS energy ratios will be output.  |

**Remarks:**

1. The CMSENRGY command may be requested in the modal solution sequences (SOLs 110, 111, 112, 145, 146, 200) and the real eigenvalue analysis solution sequences (SOLs 103 and 106). It is intended for use when superelements are defined and component modal synthesis techniques are employed. (See the MODALKE and MODALSE Case Control commands for other options.)
2. ESORT, FILTER, and TOPN describers apply only to QSET results output. TOTAL results output is always in increasing order of superelement ID number.
3. QSET CMS energy ratios are output in increasing order of component mode number unless the ESORT keyword specifies a particular sorting order. If a sorting order is specified, the magnitude of the energy ratio is sorted. DESCEND can be used as a synonym for RATIO.

4. The FILTER keyword specifies an absolute value that is used to limit the amount of printed output produced. It is applied to the magnitude of the CMS energy ratio. If the CMS energy ratio magnitude is less than fratio for any natural mode, no output for that natural mode is produced. THRESH can be used as a synonym for FILTER.
5. In order to obtain unforced response (RESPONSE=BOTH or MODAL) output in SOL 111 and SOL 112, a subcase containing the ANALYSIS = MODES option must be present.
6. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data have no meaning.

## CSSCHD

### Aerodynamic Control Surface Schedule

Selects control system schedule information.

**Format:**

CSSCHD = n

**Example:**

CSSCHD=10

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a control system schedule that appears on a CSSCHD Bulk Data entry. |

**Remark:**

1. One or more CSSCHD Bulk Data entries can be invoked by this Case Control command.

**DATAREC**

## Data Recovery Output for p-Version Elements

Requests form and type of output for p-version elements.

Format:

$$\text{DATAREC} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right] = n$$

Example:

DATAREC=12

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of grid point results per loadcase.  |
| SORT2      | Output will be presented as a tabular listing of loadcase per grid point.  |
| n          | SID of OUTPUT Bulk Data entry to be used. Only displacements, stresses, and strains of p-version elements with identification numbers that appear on the OUTPUT Bulk Data entry with SID=n will be output (Integer > 0). |

Remarks:

1. DATAREC is processed only when an adaptive analysis is requested.
2. Only one command per subcase is allowed.
3. This information is used only for output control and does not affect the analysis.
4. Displacements, stresses, and strains will be calculated and printed only for p-version elements on OUTPUT Bulk Data entries. Those elements listed that are not p-version elements will be ignored.
5. The coordinates of the view points (points at which the displacements are calculated and printed) can be printed by using the VUGRID command.

**DBSAVE****Saving Control Parameter of Datablocks for Advanced Nonlinear Elements**

Selects saving of datablocks of advanced nonlinear elements for static and transient nonlinear analysis in SOL 400.

**Format:**

DBSAVE=n

**Example:**

```
DBSAVE =2
SUBCASE=1
    ANALYSIS=HSTAT
SUBCASE=2
    ANALYSIS=NLSSTAT
```

```
DBSAVE = -1
SUBCASE=1
    STEP = 1
    ANALYSIS=NLSSTAT
DBSAVE = 1
STEP = 2
ANALYSIS=NLSSTAT

SUBCASE=2
    ANALYSIS=NLSSTAT
SUBCASE=3
    ANALYSIS=NLSSTAT
SUBCASE=4
    ANALYSIS=NLSSTAT
```

| Descriptor | Meaning   |
|------------|---|
| n          | selected value to control saving of datablocks of advanced nonlinear elements (Integer >= -1).(Default=0) |
| -1         | No datablocks of advanced nonlinear elements is saved   |
| 0          | Saving datablocks of advanced nonlinear elements at the end of each loadcase                              |
| >0         | Saving datablocks of advanced nonlinear elements at the every nth output request of results               |

**Remarks:**

1. DBSAVE can be applied above all the subcases, within subcases, and steps. DBSAVE above all the subcases is going to be applied to all the subcases.
2. If DBSAVE is present both above subcase and in step, the DBSAVE in step is dominant and used to control datablocks saving in this step.
3. When DBSAVE=-1, advanced nonlinear element can't be used in Linear Perturbation or other analysis step with NLIC.

## DEACTEL

### Elements to be Deactivated for SOL 400 Analysis

Indicates which DEACTEL Bulk Data entry is used to control the elements to be deactivated in a single physics job, or a particular physics pass of a coupled job. SOL 400 for NLSTATIC, and NLTRAN, as well Perturbation analyses only.

Format:

DEACTEL=N

Example:

DEACTEL=2

| Descriptor | Meaning   |
|------------|---|
| N          | ID of a matching DEACTEL Bulk Data entry specifying the elements to be deactivated. |

Remarks:

1. Usage is limited to the first subcase (or prior to it), the first step or the first substep of a particular physics pass. Elements are deactivated at the start of the job or the start of the particular physics pass and remain deactivated for the entire job.
2. Elements deactivated can not be reactivated at later stage in the analysis.

**DEFORM (Case)****Element Deformation Static Load**

Selects the element deformation set.

**Format:**

DEFORM=n

**Example:**

DEFORM=27

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of DEFORM Bulk Data entries (Integer > 0). |

**Remarks:**

1. DEFORM Bulk Data entries will not be used unless selected by the DEFORM command in the Case Control Section.
2. DEFORM is only applicable in linear statics, inertia relief, differential stiffness, and buckling problems (SOLs 101, 105, 114, and 200), and will produce a fatal message in other solution sequences.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement loads (SPC, SPCD).
4. Static, thermal, and element deformation loads should have unique identification numbers.
5. In the superelement solution sequences, if the DEFORM Case Control command is used in a cold start, it must also be specified in the restart.

## DESGLB

### Request Design Constraints at the Global Level

Selects the design constraints to be applied at the global level in a design optimization task.

**Format:**

DESGLB=n

**Examples:**

DESGLB=10

DESG=25

| Describer | Meaning  |
|-----------|--|
| n         | Set identification of a set of DCONSTR or a DCONADD Bulk Data entry identification number (Integer > 0). |

**Remarks:**

1. If used, this command must occur before the first subcase.
2. The DESGLB Case Control command is optional and invokes constraints that are to be applied independently of a particular subcase. These constraints could be based on responses that are independent of subcases (e.g., WEIGHT, VOLUME, WMPID, FRMASS, and FATIGUE). The DESGLB command must be used when applying constraints to FRMASS, WMPID and FATIGUE responses.
3. The DESGLB Case Control command can be used to invoke constraints that are not functions of DRESP1 entries; e.g., DRESP2 responses that are not functions of DRESP1 responses are subcase independent.
4. Constraints that are applied to responses that span subcases through the use of the DRSPAN command must be invoked using the DESGLB command.

**DESMOD****Design Model Name**

Assigns the design model parameter used to locate the associated datablocks for merging of two or more SOL 200 models using the MultiOpt application.

**Format:**

DESMOD=name

**Examples:**

DESMOD=FLUTTER

| Descriptor | Meaning  |
|------------|--|
| Name       | User-specified name to designate the model. (up to 8 characters) |

**Remarks:**

1. The name is an attribute of the datablocks that require merging across models in order to perform a simultaneous design.
2. The parameter is only used with MultiOpt, that merges simultaneous models.
3. A unique value of DESMOD is needed for each model invoked by MultiOpt.
4. If DESMOD is used in a SOL 200 input file that is not being used in MultiOpt, it will result in a fatal error.
5. MultiOpt supports Multi Model Optimization (MMO) and Global Optimization (GO), see [Design Sensitivity and Optimization User's Guide](#)

**DESOBJ****Design Objective**

Selects the DRESP1, DRESP2 or DRESP3 entry to be used as the design objective.

**Format:**

$$DESOBJ \left[ \begin{array}{c} MAX \\ MIN \end{array} \right] = n$$

**Examples:**

DESOBJ=10

DESO=25

| Describer | Meaning  |
|-----------|--|
| MIN       | Specifies that the objective is to be minimized.                                       |
| MAX       | Specifies that the objective is to be maximized.                                       |
| n         | Set identification number of a DRESP1, DRESP2 or DRESP3 Bulk Data entry (Integer > 0). |

**Remarks:**

1. A DESOBJ command is required for a design optimization task and is optional for a sensitivity task. Only one DESOBJ command may appear in a Case Control Section.
2. The referenced DRESPi entry must define a scalar response (e.g., WEIGHT or VOLUME).
3. If the DESOBJ command refers to a global response, such as weight, it should appear above the first subcase. If the DESOBJ command refers to a subcase-dependent response such as an element stress, it should appear in that subcase. If it refers to a subcase dependent response but is inserted above the first subcase, it will select the response from the first subcase for the objective and ignore the responses in subsequent subcases.
4. Using DREPS2 with weight factors, SOL200 can support multiple objective optimization.
5. MSC Nastran MultiOpt utility supports Multi Model Optimization (MMO) and Global Optimization (GO), see [Design Sensitivity and Optimization User's Guide](#)

## DESSUB

### Design Constraints Request at the Subcase Level

Selects the design constraints to be used in a design optimization task for the current subcase.

**Format:**

DESSUB=n

**Examples:**

DESSUB=10

DESS=25

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification of a set of DCONSTR and/or a DCONADD Bulk Data entry identification number (Integer $\geq 0$ ). |

**Remarks:**

1. A DESSUB Case Control command is required for every subcase for which constraints are to be applied. An exception to this is ‘global constraints’, which are selected by the DESGLB Case Control command.
2. All DCONSTR and DCONADD Bulk Data entries with the selected set ID will be used.
3. Constraints cannot be applied to the FRMASS, FATIGUE, or FRFTG response using the DESSUB command. Use the DESGLB command instead.

## DESVAR

### Design Variable Selection

Selects a set of DESVAR entries for the design set to be used.

Format:

$$DESVAR = \begin{bmatrix} ALL \\ n \end{bmatrix}$$

Example:

DESVAR=10

| Describer | Meaning   |
|-----------|---|
| n         | Set identification of a previously appearing SET command (Integer > 0). Only DESVAR Case Control commands with IDs that appear on this SET command will be used in the SOL 200 design task. |

Remarks:

1. Only one DESVAR Case Control command may appear in the Case Control Section and should appear above all subcase commands.
2. The DESVAR Case Control command is optional. If it is absent, all DESVAR Bulk Data entries will be used.

**DISPLACEMENT****Displacement Output Request**

Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.

Format:

$$\begin{aligned} \text{DISPLACEMENT} & \left[ \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] \left[ \begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{RALL} \end{array} \right], \right. \\ & \left. \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{CID} \end{array} \right], \left[ \begin{array}{c} \frac{\text{TM} = f}{\text{T1} = f, \text{T2} = f, \text{T3} = f} \end{array} \right], \left[ \begin{array}{c} \frac{\text{RM} = f}{\text{R1} = f, \text{R2} = f, \text{R3} = f} \end{array} \right], \right. \\ & \left. \left[ \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \\ \text{BOTH} \\ \text{NONE} \end{array} \right] \right] \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\} \\ \text{CONNECTOR} = \left[ \begin{array}{c} \text{ALL} \\ m \end{array} \right], \left[ \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \\ \text{BOTH} \\ \text{NONE} \end{array} \right] \right] \end{aligned}$$

**Examples:**

```
DISPLACEMENT=5
DISPLACEMENTS (REAL)=ALL
DISPLACEMENT (SORT2, PUNCH, REAL)=ALL
DISPLACEMENT (SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
DISPLACEMENT (PRINT, RALL, NORPRINT)=ALL
DISP (T1=1.-3, T3=1.-2) = ALL
DISP (TM=1.-3, PRINT,PLOT) = ALL
DISP (TM=1.-3,PRINT,PLOT,SORT2) = 20
DISP (CONN=23)=54
DISPLACEMENT (PLOT,PRINT,BOTH) = ALL
```

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of load, frequency, or time for each grid point.  |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

| Descriptor   | Meaning   |
|--------------|---|
| *            | The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated. |
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.  |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| PSDF         | Requests that the power spectral density function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.  |
| ATOCL        | Requests that the autocorrelation function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.   |
| CRMS         | Requests that the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.   |
| RALL         | Request that all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.   |
| RPRINT       | Writes random analysis results in the print file (Default).   |
| NORPRINT     | Disables the writing of random analysis results in the print file.  |
| RPUNCH       | Writes random analysis results in the punch file.   |
| CID          | Requests printing of output coordinate system ID in printed output file, (.f06).  |
| TM           | Translational magnitude filter. See Remark 8.   |
| T1, T2, T3   | Translational component filters. See Remark 8.  |
| RM           | Rotational magnitude filters. See Remark 8.   |
| R1, R2, R3   | Rotational component filters. See Remark 8.   |
| F            | Filter value (Real > 0.0). See Remark 8.  |

| Descriptor                | Meaning   |
|---------------------------|---|
| CONNECTOR                 | <p>A set of CWELD or CFAST elements are defined from which auxiliary grids will be determined for output post-processing for displacement display in the basic system. This command produces the following actions:</p> <p>The auxiliary point "grids" determined by the set m of connector elements specified on this entry will be appended to the set n defined on the right side of the DISP command.</p> <p>m is the identification of a connector element set defined by a previously appearing SET command.</p> <p>If the right side of the DISP command is defined as NONE, then no points will be output even if the user has defined the keyword CONN=ALL or CONN=m.</p> <p>If the right side of the DISP command is defined as ALL, then auxiliary point "grids" for all connectors will be generated regardless if CONN= is specified.</p> <p>If the user wishes to produce displacements for all the connector elements and have them appended to the set n defined on the right side of the DISP command, then to avoid having to define a previously appearing m SET command with all connectors listed, the user may define CONN=ALL.</p> |
| ALL                       | Displacements for all points will be output.  |
| n                         | Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0).  |
| NONE                      | Displacement for no points will be output.  |
| STRUCTURE,<br>FLUID, BOTH | Request eigenvector output data type for normal mode computation (SOL 103) with METHOD(COUPLED) case control. Structure displacement only, Fluid pressure only or both Structure displacement and Fluid pressure output separately. Default is NONE for both structure displacement and fluid pressure together. See Remark 10.   |

**Remarks:**

1. The defaults for SORT1 and SORT2 depend on the type of analysis:
  - a. SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis.
  - b. SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO, then the remaining

commands will also be output in SORT1 format. If SORT2 is selected in a static or frequency response solution for one or more of the commands ACCE, DISP, FORC, MPCF, OLOA, SPCF, STRA, STRE, and VELO, then the remaining commands will also be output in SORT2 format.

- c. XY plot requests forces SORT2 and overrides SORT1 requests!
  - d. If a RANDOM request occurs the output will be in SORT2 format, even if SORT1 is requested.
2. VECTOR and PRESSURE are alternate forms and are equivalent to DISPLACEMENT. In complex analysis, the pressure is ALWAYS magnitude-phase.
  3. DISPLACEMENT=NONE overrides an overall output request.
  4. The units of translation are the same as the units of length of the model. Rotations are in units of radians.
  5. Displacement results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
  6. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
  7. Note that the CID keyword affects only grid point related output, such as DISplacement, VELOCITY, ACCELERation, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid-related output request anywhere in the Case Control Section to turn on the printing algorithm.
  8. Displacement components may be selected to control filtering to reduce the amount of output produced. When magnitudes are selected, the component values are ignored. Only a single positive value for f can be supplied, and comparisons are performed in the global reference frame. Comparisons are performed after the SET intersection is performed against the domain. Selection of this option does not effect the MAXMIN(GRID) operations. Scalar comparisons are performed using the minimum of all supplied values for the filters.

For complex vectors encountered in frequency response analysis, the magnitudes TM and RM follow a derivation using a deterministic interpretation and are calculated as follows:

- For Grid Points

TM

Define

$$C1 = T1\text{real}^{**2} + T2\text{real}^{**2} + T3\text{real}^{**2}$$

$$C2 = T1\text{imag}^{**2} + T2\text{imag}^{**2} + T3\text{imag}^{**2}$$

$$C3 = T1\text{real} * T1\text{imag} + T2\text{real} * T2\text{imag} + T3\text{real} * T3\text{imag}$$

$$C4 = (C1 + C2)/2$$

$$C5 = (C1 - C2)/2$$

Then,

$$TM = C4 + \text{SQRT}(C5^{**2} + C3^{**2})$$

RM

The calculations are similar to the above, except that R1, R2 and R3 are used in place of T1, T2 and T3, respectively.

- For Scalar Points

In this case, TM and RM have the same meaning.

Define

$$\begin{aligned} C1 &= T1\text{real}^{**2} \\ C2 &= T1\text{imag}^{**2} \\ C3 &= T1\text{real} * T1\text{imag} \\ C4 &= (C1 + C2)/2 \\ C5 &= (C1 - C2)/2 \end{aligned}$$

Then,

$$TM \text{ (or RM)} = C4 + \text{SQRT}(C5^{**2} + C3^{**2})$$

- When using filters, the compound usage of the verbs PRINT, PLOT is allowed. The entries in the printed output are the entries that exceed any threshold, while the remaining entries within the SET are marked as plot to allow for postprocessing operations. When SORT2 is selected, then PRINT, PLOT must be used to allow for table transpose operations to occur. When any entry in the SORT2 format is above the threshold, all values for time or frequency will be printed for the grid point.
- Default eigenvector output format of coupled mode computation (SOL 103) is structure displacement and fluid pressure combined in one datablock per same modes DISP(PRINT) = ALL.

The eigenvector table has normal title in f06 file.

```
REAL EIGENVECTOR NO. 1
```

To get structure displacement and fluid pressure separately from coupled mode computation (with METHOD(coupled) in SOL 103), DISP(PRINT,STRUCTURE) = ALL is used for STRUCTURAL displacement only. The eigenvector table has title (STRUCTURE) in f06 file.

```
REAL EIGENVECTOR (STRUCTURE) NO. 1
```

DISP(PRINT,FLUID) = ALL is used for FLUID pressure only. The eigenvector table has the title (FLUID) in f06 file.

```
REAL EIGENVECTOR (FLUID) NO. 1
```

DISP(PRINT,BOTH) = ALL is used for both STRUCTURAL displacement and FLUID pressure but separately. Each eigenvector table includes its own title (STRUCTURE) or (FLUID) in f06 file.

```
REAL EIGENVECTOR (STRUCTURE) NO. 1
```

.....

```
REAL EIGENVECTOR (FLUID) NO. 1
```

.....

**DIVERG****Static Aeroelastic Divergence Request**

Selects the divergence parameters in a static aeroelastic divergence problem.

**Format:**

DIVERG=n

**Example:**

DIVERG=70

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a DIVERG Bulk Data entry (Integer > 0). |

**Remark:**

1. Static aeroelastic divergence analysis can be performed only in SOLs 144, 200 and 400.

**DLOAD****Dynamic Load Set Selection**

Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.

**Format:**

DLOAD=n

**Example:**

DLOAD=73

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE Bulk Data entry (Integer > 0). |

**Remarks:**

1. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.
2. TLOAD1 and TLOAD2 may be selected in a transient or frequency response problem.
3. Either an RLOAD*i* or TLOAD*i* entry (but not both) must be selected in an aeroelastic response problem. If RLOAD*i* is selected, a frequency response is calculated. If TLOAD*i* is selected, the transient response is computed by Fourier transform. When there are only gust loads (GUST Bulk Data entry), the DLOAD selects a TLOAD*i* or RLOAD*i* Bulk Data entry with zero load, along with field 3 of the GUST command.
4. The DLOAD command will be ignored if specified for upstream superelements in dynamic analysis. To apply loads to upstream superelements, refer to the LOADSET Case Control command.
5. For nonlinear dynamic analysis with SOL 400, TEMPERATURE load can't be applied by DLOAD/TEMP/TLOAD*i*. TEMP (case control) and TTEMP (Bulk data) should be used to apply the temperature load. However, for pure linear analysis, DLOAD/TEMP has to be used for temperature load.

**DRSPAN****Response Spanning Set Selection**

Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcases.

**Format:**

DRSPAN=n

**Example:**

DRSPAN=10

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a previously appearing SET command (Integer > 0). |

**Remarks:**

1. In SOL 200, DRESP2 or DRESP3 Bulk Data entries can invoke DRESP1 responses that span subcases if these DRESP1 responses have been identified using a DRSPAN Case Control command that references a SET request that identifies the DRESP1 Bulk Data entries.
2. Each DRESP1 identified must produce a scalar value.
3. The DRSPAN Case Control command must be at the subcase level, whereas the SET request can be specified above the subcase level.
4. DRESP2, or DRESP3 that SPANS subcases, cannot reference another DRESP2 and/or DRESP3.
5. DRSPAN must appear in every subcase in the file if the synthetic response is to span the subcases. The synthetic response via DRESP2 or DRESP3 must reference all DRESP1 IDs defined in SETs of DRSPAN. In DEQATN, all DRESP1 IDs should show up in the list of variables. For those DRESP1 IDs that are not needed, it can be dropped from the definition of equation(s).
6. To check the value of spanned response, set parameter P2 of DOPTPRM to a value equal to or larger than 8, see DOPTPRM for details. Note that verification requires performing hand calculation based on user-provided DEQATN for the spanned response.

**DSAPRT****Design Sensitivity Output Parameters**

Specifies design sensitivity output parameters.

**Format:**

$$DSAPRT \left[ \begin{bmatrix} FORMATTED \\ UNFORMATTED \\ NOPRINT \end{bmatrix}, \begin{bmatrix} NOEXPORT \\ EXPORT \end{bmatrix}, START = i, BY = j, END = k \right] = \begin{bmatrix} ALL \\ n \\ NONE \end{bmatrix}$$

**Examples:**

```
DSAPRT (FORMATTED, EXPORT)
DSAPRT (FORMATTED, START=FIRST, BY=3, END=LAST) = 101
DSAPRT (UNFORMATTED, START=FIRST)
DSAPRT (UNFORMATTED, EXPORT)
DSAPRT (FORMATTED, END=4) =ALL
DSAPRT (UNFORMATTED, END=SENS) =ALL
DSAPRT (NOPRINT, EXPORT)
```

| Descriptor  | Meaning  |
|-------------|--|
| FORMATTED   | Output will be presented with headings and labels.   |
| UNFORMATTED | Output will be printed as a matrix print (see description of the MATPRN module in the <a href="#">MSC Nastran DMAP Programmer's Guide</a> ).   |
| NOPRINT     | No output will be printed.   |
| EXPORT      | Output will be exported to an external binary file specified by PARAM,IUNIT.   |
| NOEXPORT    | Output will not be exported to an external binary file.  |
| START=i     | Specifies the first design cycle for output (Integer > 0 or Character: "FIRST" or "LAST"; Default = 1 or "FIRST").   |
| BY=j        | Specifies the design cycle interval for output. See Remark 2. (Integer $\geq$ 0; Default = 0).   |
| END=k       | Specifies the last design cycle for output. (Integer > 0 or Character: "FIRST", "LAST", or "SENS"; Default = "LAST")   |
| ALL         | All retained design responses (defined on DRESP1, DRESP2 and DRESP3 entries) will be output.   |
| n           | Set identification of a previously appearing SET command. Only sensitivities of retained responses with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. Only one DSAPRT may appear in the Case Control Section and it must occur with or above the first SUBCASE command.
2. Sensitivity data will be output at design cycles i, i+j, i+2j, ..., k. Note that the BY=0 implies no sensitivity results will be output at the intermediate design cycles.
3. END=SENS requests design sensitivity analysis, and no optimization will be performed.
4. If both DSAPRT and PARAM,OPTEXIT, 4, -4, or 7 are specified, then DSAPRT overrides PARAM,OPTEXIT, 4, -4, or 7. PARAM,OPTEXIT values and the equivalent DSAPRT commands are described as follows:

| OPTEXIT | Equivalent DSAPRT Command         |
|---------|-----------------------------------|
| 4       | DSAPRT(UNFORMATTED, END=SENS)     |
| -4      | DSAPRT(NOPRINT, EXPORT, END=SENS) |
| 7       | DSAPRT(UNFORMATTED, START=LAST)   |

5. The n and NONE options are not supported for UNFORMATTED output. Only the UNFORMATTED option is supported for EXPORT.
6. PARAM,DSZERO can be used to set a threshold for the absolute value of the formatted sensitivity prints.
7. Design Sensitivity analysis is never performed following a discrete design optimization. Therefore, no sensitivity output will be produced with DSAPRT(END=LAST) when discrete optimization is performed at the end of a job.
8. Formatted sensitivity data can also be written into Comma Separated Values(or CSV) file with following steps
  - a. DSAPRT(formatted,..) request in case control. Note that 'formatted' is a default option.
  - b. PARAM,XYUNIT,52 in bulk data. Unit 52 is simply chosen as an example.
  - c. file assignment statement, such as  
 ASSIGN USERFILE='jobname.csv' FORM=formatted STATUS=new UNIT=52

**DSYM****Dihedral Symmetry Option in Cyclic Symmetry**

Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.

Format:

$$DSYM = \left\{ \begin{array}{l} S \\ A \\ SS \\ SA \\ AS \\ AA \end{array} \right\}$$

**Example:**

DSYM=AS

| Descriptor        | Meaning  |
|-------------------|--|
| S, A              | The problem is assumed to be symmetrical (S) or antisymmetrical (A) with respect to the plane containing side 1 of segment 1.  |
| SS, SA,<br>AS, AA | The problem is assumed to be symmetrical (or antisymmetrical) with respect to the plane containing side 1 of segment 1 (denoted by the first symbol), and also with respect to a plane perpendicular to side 1 (denoted by the second symbol). |

**ECHO****Bulk Data Echo Request**

Controls echo (i.e., printout) of the Bulk Data.

Format:

$$ECHO = \left\{ \begin{array}{ll} \left[ \begin{array}{l} SORT([EXCEPT]cdni,...) \\ UNSORT \\ BOTH \\ NONE \end{array} \right], PUNCH \left[ \begin{array}{l} SORT \\ BOTH \\ NEWBULK \end{array} \right], FILE \end{array} \right\}$$

Examples:

ECHO=NOSORT  
 ECHO=BOTH  
 ECHO=PUNCH, SORT (MAT1, PARAM)  
 ECHO=SORT (EXCEPT DMI, DMIG)  
 ECHO=BOTH, PUNCH, FILE

(In the above examples See Remark 7.: the PUNCH keyword will always result in the PUNCH file containing the entire bulk data input. Explicit inclusion indicators (CDNI) such as (MAT1, PARAM) will always cause a F06 file to be written that contains only the (CDNI) data.)

| Descriptor | Meaning  |
|------------|--|
| SORT       | The sorted (arranged in alphanumeric order) Bulk Data will be printed.   |
| EXCEPT     | Excludes cdni Bulk Data entries from sorted echo printout. See Remark 6.   |
| cdni,...   | Defines Bulk Data entry names to be included, or excluded by EXCEPT, in the sorted echo printout. The PUNCH (.pch) file is not affected by cdni.                                 |
| UNSORT     | The unsorted Bulk Data will be printed. If SORT is also not specified, the sorted Bulk Data will not be printed.   |
| BOTH       | Both sorted and unsorted Bulk Data will be printed. This is equivalent to ECHO=SORT, UNSORT.   |
| NONE       | Neither sorted nor unsorted Bulk Data will be printed.   |
| PUNCH      | The entire Bulk Data will be written to the punch (.pch) file. See Remark 7. for options.  |
| FILE       | The entire Bulk Data echo will be written to the separate file with a default suffix of .BECHO in *.f06 form. A user-defined filename must be specified in the ASSIGN statement. |
| NEWBULK    | In SOL 200, a complete unsorted Bulk Data file is written to the PUNCH file with updated design model entries.   |

**Remarks:**

1. If no ECHO command appears, sorted Bulk Data will be printed.
2. Comments will appear at the front of the sorted file if ECHO=PUNCH.
3. Portions of the unsorted Bulk Data can be selectively echoed by including the commands ECHOON and ECHOOFF at various places within the Bulk Data. ECHOOFF stops the unsorted echo until an ECHOON command is encountered. Many such pairs of commands may be used. The ECHOON and ECHOOFF command may be used in the Executive and Case Control Sections; however, ECHOOFF should not be the first entry and continuation entries are not handled correctly.
4. If the SORT (cdni,...) describer is specified in a restart in SOLs 101 through 200, then the continuation entries will not be printed.
5. If the SORT (cdni,...) describer is used, then it must appear as the last describer, as in the preceding example.
6. If the EXCEPT describer is specified, then it must be specified before all cdni. All Bulk Data entry types will be listed except those given for cdn1, cdn2, etc. If EXCEPT is not specified, then only those Bulk Data entry types listed under cdn1, cdn2, etc. will be listed.
7. PUNCH without options or PUNCH(SORT) produces a sorted listing of the entire bulk data input in the punch file. PUNCH(NEWBULK) produces unsorted bulk data input with updated design model entries in the punch file. PUNCH(BOTH) combines these two outputs.

**EDE****Element Energy Loss Per Cycle Output Request**

Requests the output of the energy loss per cycle in selected elements.

Format:

$$\text{EDE} \left[ \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \left[ \begin{array}{c} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] [\text{THRESH} = p][\text{NOPERCENT}] \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

EDE=ALL

EDE (PUNCH, THRESH=.0001)=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|           |   |
|-----------|---|
| AVERAGE   | Requests average energy (in frequency response analysis only) (Default).  |
| AMPLITUDE | Requests amplitude of energy (in frequency response analysis only).   |
| PEAK      | Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.  |
| THRESH    | Energies for elements having an energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).   |
| NOPERCENT | Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will appear as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized. |
| ALL       | Energy for all elements will be computed.   |

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EDE command, or above all subcases (Integer > 0). |
| NONE       | Element energy loss will not be output.  |

#### Remarks:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by AVERAGE:

$$E_o = \pi\omega(\{u_r\}^T [B_e]\{u_r\} + \{u_i\}^T [B_e]\{u_i\})$$

#### AMPLITUDE:

$$E_a = \pi\omega\sqrt{(\{u_r\}^T [B_e]\{u_r\} - \{u_i\}^T [B_e]\{u_i\})^2 + (2\{u_r\}^T [B_e]\{u_i\})^2}$$

#### PEAK:

$$E_{peak} = E_o + E_a$$

where:

- E = elemental energy.
- $\{u_r\}$  = displacement (real part).
- $\{u_i\}$  = displacement (imaginary part).
- $[B_e]$  = elemental damping.

5. In SOLs 111 and 112, EDE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Only damping from the viscous dampers (e.g., CVISC, CDAMPi, etc.) are included. Structural damping is not included in the calculation.
7. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
8. If there are any frequency dependent CBUSH elements present then all CBUSH elements will be excluded from the element energy loss output.

9. For transient response, EDE is computed in a quasi-static manner at each time step:  $\frac{1}{2}\{v\}^T [B_e]\{u\}$  ; where  $B_e$  is elemental damping and  $v$  and  $u$  are velocity and displacement respectively.

**EKE****Element Kinetic Energy Output Request**

Requests the output of the kinetic energy in selected elements.

**Format:**

$$EKE \left[ \begin{array}{c} \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right] \left[ \begin{array}{c} AVERAGE \\ AMPLITUDE \\ PEAK \end{array} \right] [THRESH = p][NOPERCENT] \end{array} \right] = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

EKE=ALL

EKE (PUNCH, THRESH=.0001)=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|           |   |
|-----------|---|
| AVERAGE   | Requests average energy (in frequency response analysis only) (Default).  |
| AMPLITUDE | Requests amplitude of energy (in frequency response analysis only).   |
| PEAK      | Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.  |
| THRESH    | Kinetic energies for elements having a energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).  |
| NOPERCENT | Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will be appears as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized. |
| ALL       | Kinetic energy for all elements will be computed.   |

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EKE command, or above all subcases (Integer > 0). |
| NONE       | Kinetic energy values will not be output.  |

**Remarks:**

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy values may be computed in one of three ways as selected by

AVERAGE:

$$E_o = \frac{1}{4}(\{\boldsymbol{v}_r\}^T [\boldsymbol{M}_e] \{\boldsymbol{v}_r\} + \{\boldsymbol{v}_i\}^T [\boldsymbol{M}_e] \{\boldsymbol{v}_i\})$$

AMPLITUDE:

$$E_a = \frac{1}{4} \sqrt{(\{\boldsymbol{v}_r\}^T [\boldsymbol{M}_e] \{\boldsymbol{v}_r\} - \{\boldsymbol{v}_i\}^T [\boldsymbol{M}_e] \{\boldsymbol{v}_i\})^2 + (2 \{\boldsymbol{v}_r\}^T [\boldsymbol{M}_e] \{\boldsymbol{v}_i\})^2}$$

PEAK:

$$E_{peak} = E_o + E_a$$

where:

E = elemental energy.

$\{\boldsymbol{v}_r\}$  = velocity (real part).

$\{\boldsymbol{v}_i\}$  = velocity (imaginary part).

$[\boldsymbol{M}_e]$  = elemental mass.

5. In SOLs 111 and 112, EKE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
7. EKE output request is not available for linear perturbation analysis.
8. If there are any frequency dependent CBUSH elements present then all CBUSH elements will be excluded from the element kinetic energy output.

9. For transient response, EKE is computed in a quasi-static manner at each time step: $\frac{1}{2}\{v\}^T[M_e]\{v\}$  ;  
where  $M_e$  is elemental mass and  $v$  is velocity.

**ELAFORCES****Elastic Forces Output Request**

Requests the form and type of elastic forces output.

**Format:**

$$\text{ELAFORCES} \left[ \left( \begin{matrix} \text{[SORT1]} \\ \text{[SORT2]} \end{matrix} \right), \left( \begin{matrix} \text{[PRINT, PUNCH]} \\ \text{[PLOT]} \end{matrix} \right), \left( \begin{matrix} \text{[REAL or IMAG]} \\ \text{[PHASE]} \end{matrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

ELAFORCE=ALL  
 ELAF(REAL, PUNCH, PRINT)=17  
 ELAFORCE=25  
 ELAFORCE(SORT2, PRINT)=20

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |   |
|--------------|---|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.  |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| ALL          | Elastic forces for all points will be output. See Remarks 2.  |
| NONE         | No elastic force will be output.  |
| n            | Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. Refrain from using ALL in transient analysis which may produce voluminous output.
2. See Remark 1. under [DISPLACEMENT \(Case\)](#), for a discussion of SORT1 and SORT2.
3. ELAFORCE=NONE suppresses the generation of elastic forces output.
4. Although ELAFOR selects the desired grid IDs, it is to be noted that the connections should be included in the computation. Hence,  $F_i$  of a point in the requested set can be also expressed as

$$F_i = \sum_{j=1}^n K_{ij} u_j$$

5. ELAFORCE is available for SOL 108, 109, 111, 112 and SOL 200 with ANALYSIS=DFREQ, MFREQ and MTRAN. Note that ELAFORCE is available for output only and can not be utilized as design response.
6. ELAFORCE does not support external SE via OTM due to limitation.
7. ELAFORCE does not support DOMAINsolver with PARTOPT=grid.

**ELSDCON****Element Stress Discontinuity Output Request**

Requests mesh stress discontinuities based on element stresses.

**Format:**

$$ELSDCON \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \begin{Bmatrix} ALL \\ n \\ NONE \end{Bmatrix}$$

**Examples:**

ELSDCON=ALL

ELSDCON=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PUNCH            | X                   |                   | X*                   |
| PLOT             |                     | X                 | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL              Stress discontinuity requests for all SURFACE and VOLUME Case Control commands defined in the OUTPUT(POST) Section will be output.

n              Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request (Integer >0).

NONE              No element stress discontinuity output.

**Remarks:**

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME Case Control commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME Case Control commands. Also, the GPSTRESS command must be present for printed output and the STRFIELD command for postprocessing output using the .xdb file (PARAM,POST,0) for the same surfaces and volumes.

**ELSENS**

## Element Sensitivity Output for SOL 108/111

Select SOLUTION frequencies and RESPONSE DOFs for the generation element sensitivity.

Format:

$$\text{ELSENS} \left( \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix}, [\text{THRESH} = P], \text{RESPONSE} = r, \begin{bmatrix} \text{SOLUTION} = \left\{ \begin{array}{l} \text{ALL} \\ \text{self} \end{array} \right\} \\ [\text{MASS, STIFF, DYNAMIC, SQSTIFF, SQDYNA}] \end{bmatrix} \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example**

```
SET 81 = 100.0, 120.0
SET 91 = 11240/T3, 4001/T1
SET 96 = 15920      THRU 15950
$
ELSENS (RESPONSE=91, SOLUTION=81, MASS, STIFF, DYNAMIC) = 96
```

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|          |  |
|----------|--|
| THRESH   | The magnitude of element sensitivity less than p will be suppressed in all output files: print, punch, plot, .op2, and .xdb. (Default = 0.0).          |
| RESPONSE | Adjoint load response will be computed for unit load applied at grid point components in SET r.  |
| SOLUTION | Frequency responses at these forcing frequencies, defined in setf, will be used for element sensitivity computation. (Default=all forcing frequencies) |
| MASS     | Sensitivity with element mass matrices will be computed and output.  |
| STIFF    | Sensitivity with element stiffness matrices will be computed and output.   |
| DYNAMIC  | Sensitivity with element stiffness and mass matrices will be computed and output.  |
| SQMASS   | Squared sensitivity with element mass matrices will be computed and output.  |

| Descriptor | Meaning  |
|------------|--|
| SQSTIF     | Squared sensitivity with element stiffness matrices will be computed and output.   |
| SQDYNA     | Squared sensitivity with element stiffness and mass matrices will be computed and output.  |
| ALL        | Sensitivities for all elements will be calculated.   |
| n          | Set identification number. Sensitivity for all elements specified on the SET n command will be calculated. The SET n command must be specified in the same subcase as the ELSSENS command, or above all subcases ( Integer > 0 ). The IDs in set n must be EID (element ID). |
| NONE       | Elemental sensitivity will not be output.  |

**Remarks:**

1. Set r for RESPONSE has no default.
2. The equations various options of ELSSENS

$$\text{ELSENS(MASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r]$$

$$\text{ELSENS(STIFF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSEND(DYNAMIC)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSENS(SQMMASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [A_{\text{setf}}]^{*t} [M_{\text{elem}}]^* [A_r]^*$$

$$\text{ELSENS(SQSTIF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + [U_{\text{setf}}]^{*t} [K_{\text{elem}}]^* [U_r]^*$$

$$\begin{aligned} \text{ELSEND(DYNAMIC)} = & [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + \\ & [A_{\text{setf}}]^{*t} [M_{\text{elem}}]^* [A_r]^* + [U_{\text{setf}}]^{*t} [K_{\text{elem}}]^* [U_r]^* \end{aligned}$$

where  $[U_{\text{setf}}]$  is the displacement of SOLUTION

$[A_{\text{setf}}]$  is the acceleration of SOLUTION

$[U_r]$  is the displacement of RESPONSE

$[A_r]$  is the acceleration of RESPONSE

$[K_{\text{elem}}]$  is element stiffness matrix

$[M_{\text{elem}}]$  is element mass matrix

superscript \* means complex conjugate of the term.

**ELSUM****Element Summary Output Request**

Requests that a summary of element properties grouped by element type and/or element property type are to be printed.

**Format:**

$$\text{ELSUM}([EID, PID, BOTH, EIDSUM, PIDSUM, NSMCONT,SUMMARY]) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

ELSUM = 9

ELSUM (PID) = ALL

| Descriptor | Meaning  |
|------------|--|
| EID        | Element summary output is grouped by element type.   |
| PID        | Element summary output is grouped by element property type.  |
| BOTH       | Both EID and PID groupings are produced.   |
| EIDSUM     | Only a summary of the mass totals for the EID grouping is produced.  |
| PIDSUM     | Only a summary of the mass totals for the PID grouping is produced.  |
| NSMCONT    | Nonstructural mass contributions from NSM, NSM1, NSML, and NSML1 Bulk Data entries are identified.   |
| SUMMARY    | Only a summary of the mass totals is produced.   |
| ALL        | Element summary output for all elements  |
| n          | Set identification of a previously appearing SET command. Produces output for only those elements whose identification numbers appear in the list of this SET command. |
| NONE       | No element summary output is produced.   |

**Remarks:**

1. The ELSUM Case Control command produces a summary of properties for elements. The properties include element ID, material ID, length or thickness, area, volume, structural mass, nonstructural mass, total mass, and the product of total mass \* WTMASS. Total mass is the sum of the structural and nonstructural masses.

2. Certain element types produce only partial data. For these element types, no mass data is produced, and mass totals will not include any contributions from these element types. Mass data is computed for the following element types: CBAR, CBEAM, CBEND, CHEXA, CMASSi, CONM1, CONM2, CONROD, CPENTA, CQUAD4, CQUAD8, CQUADR, CRAC2D, CRAC3D, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, CTRIA6X, and CTUBE. The mass of elements with mid side nodes is approximated using only the geometry of the corner nodes. To get an accurate total mass, use the output from the Grid Point Weight Generator (PARAM,GRDPNT).
3. EIDSUM takes precedence over EID if both are present. Likewise, PIDSUM takes precedence over PID.
4. The ELSUM Case Control command is ignored in heat transfer solution sequences.
5. The NSMCONT describer produces various amounts of output depending upon whether the summary option is selected (SUMMARY, PIDSUM or EIDSUM requested). If NSMCONT is combined with PID, a table is produced that identifies the contribution of each NSM type Bulk Data entry to the total element nonstructural mass. If SUMMARY is included with PID and NSMCONT, an additional table is produced that identifies the mass contributions for each property type by property ID.
6. ELSUM output is only available for the PRINT option, not the PUNCH or PLOT options used in other commands.
7. ELSUM does not support advanced nonlinear elements.

## ENDSTEP

Specifies Final Analysis Step

Specifies final analysis step for SOL 700.

**Format:**

ENDSTEP = Value

**Example:**

ENDSTEP = 10000

| Descriptor | Meaning   |
|------------|---|
| Value      | Steps to end the simulation (Integer, Default = 9999999). |

**ENDTIME**

Specifies Final Analysis Time

Specifies final analysis time for SOL 700.

**Format:**

ENDTIME = Value

**Example:**

ENDTIME = 0.01

| Describer | Meaning   |
|-----------|---|
| Value     | Time in the applicable units for the model (usually seconds) (Real; Default = 1e+20). |

## ENDMODULE

Module Bulk Data section delimiter

Indicates (delimits) the end of a Module's Bulk Data section.

**Format:**

ENDMODULE

**Example:**

```
ENDMODULE  
END MODULE
```

**Remarks:**

1. A BEGIN MODULE command must appear somewhere before ENDMODULE.
2. The ENDDATA command may be used to terminate the last Module's Bulk Data section.

**ENTHALPY****Heat Transfer Enthalpy Output Request**

Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).

Format:

$$\text{ENTHALPY} \left[ \left( \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Example:

ENTHALPY=5

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of grid points for each time. |                   |                      |
| SORT2            | Output will be presented as a tabular listing of time for each grid point.  |                   |                      |
| PRINT or (blank) | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
|                  | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |      |  |
|------|--|
| ALL  | Enthalpy for all grid points will be output.   |
| n    | Set identification of previously appearing SET command. Only enthalpies of grid points with identification numbers that appear on this SET command will be output (Integer > 0). |
| NONE | Enthalpy for no grid points will be output.  |

Remark:

1. ENTHALPY=NONE is used to override a previous ENTHALPY = n or ENTHALPY = ALL command.

**EQUILIBRIUM**

## Equilibrium Force Output Request

Specifies options for equilibrium force balance output of applied loads, single point constraint forces and forces due to multi-point constraints and rigid elements.

Format:

$$\text{EQUILIBRIUM} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right] \\ \end{array} \right] = \left\{ \begin{array}{c} \text{YES} \\ \text{gid} \\ \text{NONE} \end{array} \right\}$$

Examples:

```
EQUILIBRIUM
EQUILIBRIUM = 501
```

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

YES Requests moment summation referenced to origin of basic coordinate system.

gid Requests moment summation referenced to basic system location specified by the coordinates of grid point gid.

NONE Equilibrium force balance output will not be generated.

Remarks:

1. The EQUILIBRIUM Case Control command produces a summary of the applied loads, single point forces of constraint (SPC), and multipoint/rigid body element forces of constraint (MPC), as well as a summation of these quantities. In order for the summation to represent all of the forces in the problem, these forces must be available and, therefore, the specification of an EQUILIBRIUM Case Control command causes the program to automatically compute the SPC and MPC forces. However,

if desired, the associated Case Control commands should request output. The single point forces of constraint are requested by the presence of an SPCFORCE command, and the multipoint/RBE constraint forces are requested by an MPCFORCE command. Applied loads are automatically generated by the presence of the LOAD selection Case Control command.

2. Results are always output in the basic coordinate system.
3. The EQUILIBRIUM Case Control command is applicable to Linear Static analysis (SOL101) only, and does not produce output if any superelements are present.

ERP

## Equivalent Radiated Power Panel Participation Factor Output Request

Requests the form and type of ERP panel participation factor output for frequency or transient analysis.

## Format:

$$\begin{aligned}
& ERP \left[ \begin{bmatrix} SORT2 \\ SORT1 \end{bmatrix}, \begin{bmatrix} PRINT & PUNCH \\ & PLOT \end{bmatrix} \right] \text{SOLUTION} = \left\{ \begin{array}{l} ALL \\ setf \\ sett \end{array} \right\}, \\
& \left[ MPF = \left\{ \begin{array}{l} ALL \\ nlm \\ NONE \end{array} \right\} \right], [MPFSORT = sorttype], [ELEMENT], \\
& \left[ KEY = \left\{ \begin{array}{l} frequency \\ fraction \\ time \end{array} \right\} \right], \left[ FILTER = \left\{ \begin{array}{l} 0.01 \\ real\_value \end{array} \right\} \right], \\
& \left[ ERPRHO = \left\{ \begin{array}{l} 1.0 \\ real\_value \end{array} \right\} \right], \left[ ERPC = \left\{ \begin{array}{l} 1.0 \\ real\_value \end{array} \right\} \right] \\
& \left[ RHOCP = \left\{ \begin{array}{l} 1.0 \\ real\_value \end{array} \right\} \right], \left[ ERPRLF = \left\{ \begin{array}{l} 1.0 \\ real\_value \end{array} \right\} \right] \\
& \left[ ERPREFDB = \left\{ \begin{array}{l} 1.0 \\ real\_value \end{array} \right\} \right], [CSV = unit] = \left\{ \begin{array}{l} ALL \\ setp \\ NONE \end{array} \right\}
\end{aligned}$$

Examples:

```

SET 17 = 10.,20.,30.,40.,80.,100. $ A list of frequencies
SET 25 = ROOF, DOORLF $ A list of ERP Panel names
                           $ from a ERPPNL Bulk Entry
ERP ( PRINT,PUNCH,SOLUTION=17,KEY=frac ) = 25

```

| Descriptor       | Meaning  |    |
|------------------|--|----|
| SORT1            | Output is presented as a tabular listing of ERP panels for each frequency. |    |
| SORT2            | Output is presented as a tabular listing of frequency for each ERP panel.  |    |
| PRINT or (blank) | Printer File (.f06)      Punch File (.pch)      Plot File (.op2/.h5)       |    |
| PUNCH            | X  | X* |
| PLOT             |  | X* |
|                  |  | X* |

| Descriptor | Meaning   |
|------------|---|
|            | * The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.   |
| SOLUTION   | <p>Keyword to select frequencies or time.</p> <p>ALL If associated with SOLUTION, all frequencies or times are selected. If associated with <i>setp</i>, all ERPPNL entries are selected.</p> <p>setf Identifier of Case Control SET command defining frequencies.</p> <p>sett Identifier of Case Control SET command defining times.</p>   |
| MPF        | <p>Requests the output of Modal Participation Factors for ERP. See Remark 9.</p> <p>ALL All structural modes for which MPF will be computed</p> <p>nlm Number of lowest structural modes for which MPF will be computed</p> <p>NONE MPF will not be computed. (Default)</p>   |
| MPFSORT    | <p>Keyword selecting the sort type. Default is sorting by increasing natural mode number. See Remark 10.</p> <p>Sorttype Sort options:</p> <ul style="list-style-type: none"> <li>ABSA output will be sorted by absolute value in ascending order.</li> <li>ABSD output will be sorted by absolute value in descending order.</li> <li>ALGA output will be sorted by algebraic value in ascending order.</li> <li>ALGD output will be sorted by algebraic value in descending order</li> </ul>            |
| ELEMENT    | Keyword to request element-by-element ERP output  |
| KEY        | <p>Keyword selecting the output item used to sort the printed output. The default produces output sorted on either frequency (SORT2) or ERP panel name (SORT1).</p> <p>KEY=<i>fraction</i> produces output sorted in descending order of the fractional ERP value of maximum ERP across all frequencies or time steps of a panel.</p> <p>KEY=time produces output sorted on either time (SORT2) or ERP panel name (SORT1). In transient the default KEY=frequency will automatically become KEY=time.</p> |
| FILTER     | Keyword specifying the value of a filter to be applied to the printed output only. ERP values are printed only if the fractional ERP value of maximum ERP across all frequencies or time steps of a panel exceeds the filter value.   |
| ERPRHO     | Fluid density for Equivalent Radiated Power (ERP) analysis. This item is actually an MSC Nastran parameter.   |
| ERPC       | Phase speed of the fluid for Equivalent Radiated Power (ERP) analysis. This item is actually an Nastran parameter.  |

| Descriptor | Meaning  |      |   |      |   |      |                            |
|------------|--|------|---|------|---|------|----------------------------|
| ERPRLF     | Radiation loss factor. In frequency the scale factor $C = ERPRLF * (\frac{1}{2}ERPRHO * ERPC)$ . In transient the scale factor $C = ERPRLF * (ERPRHO * ERPC)$ .  |      |   |      |   |      |                            |
| RHOCP      | Scale factor used in dB computation. This item is actually an MSC Nastran parameter.   |      |   |      |   |      |                            |
| ERPREFDB   | Scale factor used in dB computation. This item is actually an MSC Nastran parameter.   |      |   |      |   |      |                            |
| CSV        | <p>Results will be written to a .csv file.</p> <table> <tr> <td>unit</td> <td>Unit of the .csv file as used on the required ASSIGN statement.</td> </tr> <tr> <td>setp</td> <td>Identifier of case control SET command defining NAMEi entries from an ERPPNL Bulk Data entry defining panels.</td> </tr> <tr> <td>NONE</td> <td>No ERP output is produced.</td> </tr> </table> | unit | Unit of the .csv file as used on the required ASSIGN statement. | setp | Identifier of case control SET command defining NAMEi entries from an ERPPNL Bulk Data entry defining panels. | NONE | No ERP output is produced. |
| unit       | Unit of the .csv file as used on the required ASSIGN statement.  |      |   |      |   |      |                            |
| setp       | Identifier of case control SET command defining NAMEi entries from an ERPPNL Bulk Data entry defining panels.  |      |   |      |   |      |                            |
| NONE       | No ERP output is produced.   |      |   |      |   |      |                            |

#### Remarks:

1. ERP is required to produce any ERP output and ERP request must appear either above SUBCASE level or within 1st SUBCASE. Subsequent SUBCASEs, from 2nd onwards, may or may not have ERP request.
2. Output is generated in SORT2 by default. Unlike other Case Control requesting SORT2 format, the ERP command does not force all other output into SORT2 format.
3. FILTER has no effect on PUNCHed, CSV or OP2 output.
4. In addition to individual panel output a summary named ALLPANEL is produced. If there are multiple subcases, the panel name is formed from the serial subcase number (1-nsubc) and the characters 'ALLP' as in ALLP0002 unless the ERP command request output for ALL panels across the Subcases. In this case, the summary panel name ALLPANEL is retained.
5. Selectable frequencies are dependent on the presence of an OFREQ Case Control command.
6. ERPRHO, ERPC, ERPRLF, RHOCP, and ERPREFDB are actually PARAM,name,value entries.
7. The filter process avoids printing ERP for cases where ERP/ERPMAX is less than the FILTER value. ERPMAX is the maximum ERP value across all frequencies for a panel.
8. If output to a .csv file is requested, the file must be assigned with logical key USERFILE and FORM=FORMATTED, e.g.,
 

```
ASSIGN USERFILE = myfile.csv UNIT=50 FORM=FORMATTED STATUS=NEW
```
9. The MPF of ERP will be computed with the same parameters, such as SOLUTION, csv, setp and etc.
10. MPFs for ERP are sorted by increasing order of natural frequencies unless the MPFSORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. Since ERP is always positive, ABSA/ALGA and ABSD/ALGD will produce same sorting results.

11. Element ERP includes elements defined via ERPPNL and SET3. For 3D element types, such as HEXA, PENTA and TETRA, element ERP includes those elements at the outer surface of the model. For an element with multiple faces exposed, the element ERP is the summation of ERP of all exposed faces.
12. MPF for ERP and element ERP may increases the volume of output significantly.

**ESE****Element Strain Energy Output Request**

Requests the output of the strain energy in selected elements.

**Format:**

$$ESE \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right] \left[ \begin{array}{c} AVERAGE \\ AMPLITUDE \\ PEAK \end{array} \right] [THRESH = p][NOPERCENT] ) = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

**Examples:**

ESE=ALL

ESE (PUNCH, THRESH=.0001)=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|           |   |
|-----------|---|
| AVERAGE   | Requests average energy in frequency response analysis only.  |
| AMPLITUDE | Requests amplitude of energy in frequency response analysis only.   |
| PEAK      | Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.  |
| THRESH    | Energies for elements having an energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).   |
| NOPERCENT | Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will appear as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized. |
| ALL       | Energy values for all elements will be computed.  |

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the ESE command, or above all subcases (Integer >0). |
| NONE       | Element strain energy will not be output.   |

**Remarks:**

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element strain energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by AVERAGE:

$$E_o = \frac{1}{4}(\{u_r\}^T [K_e] \{u_r\} + \{u_i\}^T [K_e] \{u_i\})$$

**AMPLITUDE:**

$$E_a = \frac{1}{4} \sqrt{(\{u_r\}^T [K_e] \{u_r\} - \{u_i\}^T [K_e] \{u_i\})^2 + (2 \{u_r\}^T [K_e] \{u_i\})^2}$$

**PEAK:**

$$E_{peak} = E_o + E_a$$

where:

E = elemental energy.

$\{u_r\}$  = displacement (real part).

$\{u_i\}$  = displacement (imaginary part).

$[K_e]$  = elemental stiffness.

5. In SOLs 111 and 112, ESE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Element data recovery for thermal loads is not currently implemented in dynamics.
7. Element strain energy is available for nonlinear static analysis (SOL 106). However, in a normal modes analysis in SOL 106 with PARAM,NMLOOP (or ANALYSIS=MODES) or a PARAM NMLOOP restart into SOL 103, energies are computed for elements with linear properties only. All other nonlinear solution sequences do not support element strain energy output.

8. The strain energy for nonlinear elements is calculated by integrating the specific energy rate, the inner product of strain rate, and stress over element volume and time.

$$E = \int_0^t \int_V \dot{\varepsilon}^T \sigma dV d\tau \quad (5-10)$$

where:

- $\sigma$  = stress tensor
- $\dot{\varepsilon}$  = tensor of the strain rate
- $V$  = element volume
- $t$  = actual time in the load history

Loads from temperature changes are included in Eq. (5-10). If we assume a linear variation of temperatures from subcase to subcase, then the strain energy in Eq. (5-10) for the special case of linear material and geometry becomes

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et} \quad (5-11)$$

where  $P_{et}$  is the element load vector for temperature differences.

For linear elements, the default definition of element strain energy is

$$E = \frac{1}{2} u^T K_e u - u^T P_{et} \quad (5-12)$$

where  $P_{et}$  is the element load vector for temperature differences and element deformation.

In Eq. (5-12), the temperatures are assumed to be constant within a subcase. The default definition of the strain energy for linear elements differs from the definition for nonlinear elements by a factor of 1/2 in the temperature loads. To request the strain energy for linear elements using Eq. (5-11), set the parameter XFLAG to 2; the default value for XFLAG is 0, which uses Eq. (5-12) for the strain energy of linear elements.

9. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
10. The advanced elements are not supported in ESE.
11. If there are any frequency dependent CBUSH elements present then all CBUSH elements will be excluded from the element strain energy output.
12. For transient response, ESE is computed in a quasi-static manner at each time step:  $\frac{1}{2} u^T K_e u$ ; where  $K_e$  is elemental stiffness and  $u$  is displacement.

**EXPORTLD****Saves a Load Vector on the Database for Subsequent Reuse**

Saves the load vector for the current subcase on the database.

**Format:**

$$\text{EXPORTLD}[(\text{LOADID} = lid)(\text{LOADNAME} = Idname)]) = \left\{ \begin{array}{l} n \\ \text{ALL} \end{array} \right\}$$

**Examples:**

EXPO (LOADNAME=LANDGEAR)  
EXPORTLD=10

| Descriptor      | Meaning  |
|-----------------|--|
| LOADID=lid      | User assignable value for the LOADID descriptor. Used to uniquely identify a saved load vector for later retrieval (using DBLOCATE, for example). The default value is the subcase ID. |
| LOADNAME=Idname | User-assignable value (of up to eight characters) for the LOADNAME descriptor. Used to uniquely identify a saved load vector for later retrieval. The default value is blank.          |
| N               | Results for grid point components in SET n will be exported.   |
| ALL             | Results for ALL grid point components will be exported. Default is ALL.  |

**Remarks:**

1. Each load vector is stored individually as a one column matrix that is qualified by LOADID and LOADNAME. The combination of both the LOADID and the LOADNAME descriptors should uniquely identify the load vector to avoid overwriting (or possibly triggering the output twice rule an existing load vector).
2. If placed above the SUBCASE entry, then the load vectors for all subcases are saved.
3. The load vectors are qualified by LOADID and LOADNAME for selection using the WHERE clause on FMS commands that support it.
4. The following table shows the value of the LOADID and LOADNAME descriptors that are assigned for various EXPORTLD requests:

| Example                             | Results  |
|-------------------------------------|--|
| EXPORTLD                            | LOADID=subcase ID, LOADNAME=' - default value applied. |
| EXPORTLD(LOADID=12)                 | LOADID=12, LOADNAME=' '                                |
| EXPORTLD(LOADID=1,LOADNAME=FORCE12) | LOADID=1, LOADNAME='FORCE12'                           |
| EXPORTLD(LOADNAME=ALLCASES)         | LOADID=subcase ID,<br>LOADNAME='ALLCASES'              |

5. The load vector is typically imported into a run using the FMS DBLOCATE statement. The imported load is referenced by using its LOADID value on a LOAD Case Control command or Bulk Data entry. For example:

```

ASSIGN loads1='run1.MASTER'
DBLOCATE datablk=(EXTLD) WHERE (LOADNAME='ALLCASES'),
CONVERT (LOADID=LOADID+1000) LOGICAL=loads1
...
CEND
LOADS=1001 $ Select external load with LOADID=1001, imported from
previous run.

```

**EXTDRIN****External Superelement Data Recovery Restart Run Request**

Requests the job to perform an external superelement data recovery restart. Also specifies the storage media of the boundary solution data.

**Format:**  
**EXTDRIN** 
$$\left[ \begin{array}{c} DMIGOP2 [= \text{unitop2}] \\ MATRIXOP4 \text{ or } MATOP4 [= \text{unitop4}] \\ \boxed{\text{MATRIX or MATDB}} \\ DMIGDB \end{array} \right]$$

**Examples:**

EXTDRIN  
 EXTDRIN (DMIGOP2=45)  
 EXTDRIN (MATOP4)

| Descriptor                              | Meaning   |
|---|---|
| DMIGOP2 = <i>unitop2</i>                | Retrieve the boundary solution data stored in DMIG Bulk Data internal format on an .op2 file whose Fortran unit number is given by unitop2 (Integer > 0, Default=31). |
| MATRIXOP4 or<br>MATOP4 = <i>unitop4</i> | Retrieve the boundary solution data on an .op4 file whose Fortran unit number is given by unitop4. (Integer > 0, Default=31).   |
| MATRIXDB or<br>MATDB                    | Retrieve the boundary solution data stored in the standard matrix format from the database.   |
| DMIGDB                                  | Retrieve the boundary solution data stored in the DMIG Bulk Data internal format from the database.   |

**Remarks:**

1. EXTDRIN is intended for step three in external superelement analysis. External superelement data recovery is accomplished by restarting from the data base created in step one (external superelement creation run) and attaching the boundary solution data from step two (assembly run). The boundary solution data must have been stored in step two via the EXTDROUT Case Control command or user parameter EXTDROUT.
2. External superelement data recovery restarts are limited to SOLs 101, 103, 107 through 112, and 400.
3. EXTDRIN must be specified above all subcases.
4. User parameters EXTDR and EXTDRUNT are the old method for requesting an external superelement data recovery restart. EXTDRIN Case Control command takes precedence over the EXTDR and EXTDRUNT user parameters.
5. If data recovery is desired for the external component in SOLs 101, 103, 107 through 112 and 400, there are four methods to transmit the displacements of the reduced model, in Step 2, to the external full model. The method is selected by the EXTDROUT Case Control command in Step 2. The options are as follows:

- a. EXTDROUT(MATRIXDB[=dbext]). The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model. Keyword dbext exports the boundary solution data to a new DBset initialized by the “INIT dbext” statement in the File Management section of the input.
- b. EXTDROUT(DMIGDB[=dbext]). The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. Keyword dbext exports the boundary solution data to a new DBset initialized by the “INIT dbext” statement in the File Management section of the input.
- c. EXTDROUT(DMIGOP2=unit). The same as EXTDROUT(DMIGDB) except that the displacements of the reduced model are written in OUTPUT2 format to a tape unit specified by the unit keyword (Default=31). The output unit can be assigned to a specific file by using an ASSIGN OUTPUT2 command in the File Management Section.
- d. EXTDROUT(MATOP4=unit). The same as EXTDROUT(MATRIXDB) except that the displacements of the reduced model are written in OUTPUT4 format to a tape unit specified by the unit keyword (Default=31). The output unit can be assigned to a specific file by using an ASSIGN OUTPUT4 command in the File Management Section.

Data recovery restart for the external superelement (Step 3) is available in SOLs 101, 103, 107 through 112, and 400 and is accomplished by using a restart procedure from the data base created in Step 1 and specifying the EXTDRIN Case Control command to import the solutions from Step 2. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The setups for Steps 2 and 3 are:

| EXTDROUT Option     | Step-2 Assembly Job   | Step-3 Data Recovery for External SE  |
|---------------------|---|---|
| MATRIXDB, or DMIGDB | <p>If <i>dbext</i> is specified in EXTDRROUT then in FMS:</p> <p>INIT <i>dbext</i></p> <p>In Case Control:</p> <p>EXTDRROUT (MATRIXDB (or DMIGDB) [=<i>dbext</i>])</p> <p>On command line run with scr=no assuming job name is called 'step2.dat'</p> | <p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE (SEID=YYY), LOGICAL=SEYYY</pre> <p>In Case Control:</p> <pre>EXTDRIN (MATRIXDB or DMIGDB)</pre> |
| DMIGOP2             | <p>In FMS:</p> <pre>ASSIGN OUTPUT2='extsdr.op2', UNIT=45</pre> <p>In Case Control:</p> <pre>EXTDRROUT (DMIGOP2=45)</pre>  | <p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN INPUTT2='extsdr.op2', UNIT=45</pre> <p>In Case Control:</p> <pre>EXTDRIN (DMIGOP2=45)</pre>  |
| MATOP4              | <p>In FMS:</p> <pre>ASSIGN OUTPUT4='extsdr.op4', UNIT=45</pre> <p>In Case Control:</p> <pre>EXTDRROUT (MATOP4=45)</pre>   | <p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN INPUTT4='extsdr.op4', UNIT=45</pre> <p>In Case Control:</p> <pre>EXTDRIN (MATOP4=45)</pre>   |

6. For SOL 101, the Step 3 Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in step one to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.
7. For SOLs 103 and 107 through 112, the Step 3 Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, FREQ/DLOAD or TSTEP/DLOAD subcases.

8. SOL 400 with multidisciplinary and linear perturbation analyses will produce multiple boundary solutions in Step 2; i.e., a boundary solution for each SUBCASE with a unique ANALYSIS command. Hence in SOL 400 for Step 3 as soon as the boundary solutions have been imported, a summary of the boundary solutions will be printed in the f06. Here is a sample:

| Assembly Run Solutions |               |         |      |              |   |
|------------------------|---------------|---------|------|--------------|---|
| Solution ID            | Analysis type | Subcase | Step | Superelement | Title/Subtitle/Label  |
| 1                      | STATICS       | 2       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>STATICS   |
|                        |               |         |      |              | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>STATICS FOR BUCKLING ANALYSIS                                   |
| 2                      | MODES         | 7       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>NORMAL MODES - SOLVE ALL EIGENVALUES                            |
| 3                      | BUCKLE        | 6       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>BUCKLING - SOLVE ALL EIGENVALUES                                |
| 4                      | MODES         | 3       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>MODAL FREQUENCY - FOUR FREQUENCIES AND DELTA-F=.001 AND F1=1.04 |
| 5                      | MFREQ         | 3       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>MODAL FREQUENCY - FOUR FREQUENCIES AND DELTA-F=.001 AND F1=1.04 |
| 6                      | MODES         | 1       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001             |
| 7                      | MTRAN         | 1       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001             |
| 8                      | MCEIG         | 4       | 0    | 100          | MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS<br>HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED<br>MODAL COMPLEX - SOLVE FOR FIRST FIVE EIGENVALUES                |

The “Solution ID” column contains an arbitrary identification number which may be specified on the SOLUTION Case Control command in Step 3 whose usage will be described below. The “Analysis Type”, “Subcase”, and “Step” columns are the same as the ANALYSIS, SUBCASE, and STEP commands specified on Step 2. The “Superelement” column pertains the external superelement ID processed in Step 2. The “Title/Subtitle/Label” is taken from the TITLE, SUBTITLE, and LABEL commands specified in Step 2.

In Step 3, as with any restart, the Case Control must contain the same loads and boundary condition (LBC) commands specified in Step 1 like LOAD, SPC, MPC, METHOD, etc. If there is only one subcase specified in Step 3 then, by default, SOL 400 will perform external superelement data recovery for all solutions above. Here is a sample Step 3 Case Control setup:

```
EXTDRIN ...
SPC=1
MPC=3
LOAD=5
$
DISP=ALL
STRESS=ALL
```

The displacement and element stress request for all solutions listed in the table above will be computed.

If, however, data recovery is desired at only a few solutions then a SUBCASE and SOLUTION command pair is required to obtain data recovery at each of the desired solutions. The subcases may be specified in any order and the SOLUTION command specifies the desired “Solution ID” from the table above. For example,

```
EXTDRIN ...
SPC=1
MPC=3
SUBCASE 1
    SOLUTION=5
    STRESS=ALL
SUBCASE 2
    SOLUTION=3
    DISP=ALL
```

Alternatively, the ANALYSIS command may be used instead of the SOLUTION command. For example,

```
EXTDRIN ...
SPC=1
MPC=3
SUBCASE 1
    ANALYSIS=MFREQ
    STRESS=ALL
SUBCASE 2
    ANALYSIS=BUCKLE
    DISP=ALL
```

If there is more than one solution associated with a particular analysis type, then results will be computed for all “Solution IDs” corresponding to that analysis type.

9. In Step 2, more than one external superelement may be processed by EXTDROUT. The following must be noted if DMIGOP2 or MATOP4 is used.

- a. In SOLs 101, 103, and 107 through 112, the user must specify separate EXTDROUT as shown in the example below:

```
SUBCASE 1
    SUPER=10
    EXTDROUT (DMIGOP2=45)
SUBCASE 2
    SUPER=20
    EXTDROUT (DMIGOP2=46)
```

- b. In SOL 400, the user may specify separate EXTDROUT as described above or the same EXTDROUT command for all external superelements. If the user specifies the same EXTDROUT, then in the 3rd step a fatal message will be issued as shown below:

| Assembly Run Solutions |               |         |      |               |   |
|------------------------|---------------|---------|------|---------------|---|
| Solution ID            | Analysis type | Subcase | Step | Super-element | Title/Subtitle/Label                        |
| 1                      | MODES         | 1       | 0    | 10            | MODAL, 3-STEP EXTERNAL SE, DMIGDB, ASSEMBLY |
| 2                      | MODES         | 1       | 0    | 20            | MODAL, 3-STEP EXTERNAL SE, DMIGDB, ASSEMBLY |

\*\*\* USER FATAL MESSAGE 22907 (EXTDRSUM)

The attached boundary solutions %1 contains solutions for more than one external superelement and the desired superelement ID has not been selected.

USER ACTION: If the EXTDRROUT Case Control command (or PARAM,EXTDRROUT) specified DMIGOP2 or MATOP4 then you must specify the superelement ID on the SUPER Case Control command to select the appropriate boundary solution.

If EXTDRROUT Case Control command (or PARAM,EXTDRROUT) specified MATRIXDB or DMIGDB then you must specify the superelement ID  
 a) On the SUPER Case Control command to select the appropriate boundary solution.  
 or  
 b) In the WHERE(SEID=superelement\_ID) clause on the DBLOCATE statement of the boundary solutions database to select the appropriate boundary solution.

The summary shows that the solution file contains solutions for superelements 10 and 20 and the message indicates that only one solution may be processed. So the user must specify the SUPER command to select the desired superelement; e.g.,

```
SUPER=10
EXTDRIN (DMIGOP2=45)
```

Alternatively for EXTDRIN(DMIGDB or MATRIXDB) only, the user may specify WHERE(SEID=10) on the DBLOCATE statement of the boundary solution database.

10. The following examples continue from the examples described in Remark 18 under the EXTSEOUT Case Control command description.

#### Example 1. MATDB / MATRIXDB or DMIGDB Option

- Assembly Job (assume input file is named “assembly.dat”)
  - File Management Section (FMS) Requirement only if it is desired to store the boundary solutions on a dbset separate from DBALL

INIT EXTDRDB

- Case Control Requirement for MATDB / MATRIXDB Option.

Store boundary solutions on the DBALL dbset

EXTDROUT

Store boundary solutions on the EXTDRDB dbset

EXTDROUT ( MATRIXDB = EXTDRDB )

- Case Control Requirement for DMIGDB Option

Store boundary solutions on the DBALL dbset

EXTDROUT ( DMIGDB )

Store boundary solutions on the EXTDRDB dbset

EXTDROUT ( DMIGDB = EXTDRDB )

For all options above, “scr = no” must be specified on the Nastran command line to ensure that the databases are saved at the end of the jobs.

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)
  - File Management Section (FMS) Requirement

### Example 1. MATDB / MATRIXDB or DMIGDB Option

```

ASSIGN SE10='extse10.MASTER'
RESTART LOGI=SE10
ASSIGN ASMB='assembly.MASTER'
DBLOC DATABLK=(EXTDB) WHERE(SEID=10) LOGI=SE10

```

- Case Control Requirement for MATDB / MATRIXDB Option  
EXTDRIN
- Case Control Requirement for DMIGDB Option  
EXTDRIN ( DMIGDB )

### Example 2. DMIGOP2 Option

#### ■ Assembly Job

- Setup for SOLs 101, 103, 107 through 112, and 400 each superelement boundary solution is stored on a separate Fortran unit
- File Management Section (FMS) Requirement

```

ASSIGN OUTPUT2='extse10dr.op2' UNIT=25 DELETE
ASSIGN OUTPUT2='extse20dr.op2' UNIT=26 DELETE
ASSIGN OUTPUT2='extse30dr.op2' UNIT=27 DELETE

```

- Case Control Requirement

```

SUBCASE 1
SUPER = 10
EXTDROUT ( DMIGOP2 = 25 )
SUBCASE 2
SUPER = 20
EXTDROUT ( DMIGOP2 = 26 )
SUBCASE 3
SUPER = 30
EXTDROUT ( DMIGOP2 = 27 )

```

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT2='extalldr.op2' UNIT=25 DELETE
```

- Case Control Requirement

```
EXTDROUT ( DMIGOP2 = 25 )
```

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)

- Setup for SOLs 101, 103, 107 through 112, and 400

### Example 2. DMIGOP2 Option

- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'  
RESTART LOGICAL=SE10  
ASSIGN INPUTT2='extse10dr.op2' UNIT=25
```
- Case Control Requirement

```
EXTDRIN ( DMIGOP2 = 25 )
```
- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit
- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'  
RESTART LOGICAL=SE10  
ASSIGN INPUTT2='extalldr.op2' UNIT=25
```
- Case Control Requirement – SUPER command is required to select the correct superelement boundary solution

```
SUPER = 10  
EXTDRIN ( DMIGOP2 = 25 )
```

### Example 3. MATOP4 Option

- Assembly Job
  - Setup for SOLs 101, 103, 107 through 112, and 400 each superelement boundary solution is stored on a separate Fortran unit.
  - File Management Section (FMS) Requirement

```
ASSIGN OUTPUT4='extse10dr.op4' UNIT=25 DELETE  
ASSIGN OUTPUT4='extse20dr.op4' UNIT=26 DELETE  
ASSIGN OUTPUT4='extse30dr.op4' UNIT=27 DELETE
```
  - Case Control Requirement

```
SUBCASE 1  
SUPER = 10  
EXTDROUT ( MATOP4 = 25 )  
SUBCASE 2  
SUPER = 20  
EXTDROUT ( MATOP4 = 26 )  
SUBCASE 3  
SUPER = 30  
EXTDROUT ( MATOP4 = 27 )
```
  - Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit.

### Example 3. MATOP4 Option

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT4='extalldr.op4' UNIT=25 DELETE
```

- Case Control Requirement

```
EXTDROUT ( MATOP4 = 25 )
```

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)

- Setup for SOLs 101, 103, 107 through 112, and 400

- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'
```

```
RESTART LOGICAL=SE10
```

```
ASSIGN INPUTT4='extse10dr.op4' UNIT=25
```

- Case Control Requirement

```
EXTDRIN ( MATOP4 = 25 )
```

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit

- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'
```

```
RESTART LOGICAL=SE10
```

```
ASSIGN INPUTT4='extalldr.op4' UNIT=25
```

- Case Control Requirement – SUPER command is required to select the correct superelement boundary solution

```
SUPER = 10
```

```
EXTDRIN ( MATOP4 = 25 )
```

## EXTDROUT

External Superelement Boundary Solution Storage Request.

Requests the job to store external superelement boundary displacements and column labels; e.g., eigenvalues, forcing frequencies, time steps. Also specifies the storage media of the boundary solution data.

Format:  
**EXTDROUT**  $\left[ \begin{array}{l} DMIGOP2 [= unitop2] \\ MATRIXOP4 or MATOP4 [= unitop4] \\ \hline MATRIXDB or MATDB [= dbext] \\ DMIGDB [= dbext] \end{array} \right] NOCASE$

Examples:

**EXTDROUT**

**EXTDROUT (DMIGOP2=45)**

**EXTDROUT (MATOP4 NOCASE)**

| Descriptor                               | Meaning  |
|--|--|
| <b>DMIGOP2 = unitop2</b>                 | Store the boundary solution data in DMIG Bulk Data internal format on an .op2 file whose Fortran unit number is given by unitop2 (Integer > 0, Default=31)..   |
| <b>MATRIXOP4 or<br/>MATOP4 = unitop4</b> | Store the boundary solution data on an .op4 file whose Fortran unit number is given by unitop4. (Integer ≠ 0, Default=31). unitop4 is a non-zero integer with the following meanings:<br><br><i>unitop4</i> > 0: Store in sparse format<br><br><i>unitop4</i> < 0: Store in non-sparse format on Fortran unit number given by  unitop4 |
| <b>MATRIXDB or<br/>MATDB = dbext</b>     | Store the boundary solution data in the standard matrix format in the database. Dbext is the name of a new dbset on which to store the boundary data. Dbext must be allocated on an INIT FMS statement.  |
| <b>DMIGDB</b>                            | Store the boundary solution data in the DMIG Bulk Data internal format in the database. Dbext is the name of a new dbset on which to store the boundary data. Dbext must be allocated on an INIT FMS statement.  |
| <b>NOCASE</b>                            | NOCASEBy default in SOL 400 only, the TITLE/SUBTITLE/LABEL contents are exported along with boundary data. Specify NOCASE if you do not want these contents to be exported.  |

Remarks:

1. EXTDROUT is intended for step two (or assembly run) in external superelement analysis.
2. EXTDROUT is honored in SOLs 101, 103, 107 through 112, and 400 only.

3. In SOLs 101, 103, and 107 through 112, if there are multiple external superelements and DMIGOP2 or MATOP4 is requested then EXTDROUT must be specified in separate subcases for each external superelement along with the SUPER command and unitop4 or unitop2 must be unique for each external superelement. Separate EXTDROUT commands are not required in SOL 400. See Remarks 9. and 10. under the EXTDRIN Case Control command description.
4. User parameters EXTDROUT and EXTDRUNT are the old method for requesting the storage of boundary solution data. EXTDROUT Case Control command takes precedence over the EXTDROUT and EXTDRUNT user parameters.
5. See EXTDRIN's case control command description for a description of EXTDROUT and EXTDRIN usage.

**EXTSEOUT**

## External Superelement Creation Specification

Format:

$$\begin{aligned}
 EXTSEOUT & \left[ \begin{array}{ccccccc} STIFFNESS & MASS & DAMPING & K4DAMP & LOADS & FSCOUP, \\ ASMBULK = \begin{cases} MAN \\ MANQ \\ AUTO \end{cases}, EXTBULK, EXTID = seid, \begin{bmatrix} QSET \\ ASET \end{bmatrix}, \right. \\
 DMIGSFIX = \begin{cases} cccccc \\ EXTID \end{cases} & \left. \begin{bmatrix} MATDB \text{ (or MATRIXDB)} [= dbext] \\ DMIGDB [= dbext] \\ DMIGOP2 = unitop2 \\ DMIGPCH \\ MATOP4 \text{ (or MATRIXOP4)} = unitop4 \end{bmatrix} \right) \right]
 \end{aligned}$$

Example(s):

```

EXTSEOUT
EXTSEOUT (DMIGOP2=26)
EXTSEOUT (ASMBULK EXTID=200)
EXTSEOUT (ASMBULK EXTBULK EXTID=100)
EXTSEOUT (ASMBULK=AUTO EXTBULK EXTID=100)
EXTSEOUT (ASMBULK=MANQ EXTID=10 DMIGDB)
EXTSEOUT (ASMBULK EXTID=100 DMIGOP2=26)
EXTSEOUT (ASMBULK EXTID=100 DMIGPCH)
EXTSEOUT (ASMBULK EXTID=100 DMIGSFIX=XSE100 DMIGPCH)
EXTSEOUT (ASMBULK EXTID=200 DMIGSFIX=EXTID DMIGPCH)
EXTSEOUT (ASMBULK EXTID=100 MATOP4=26)
EXTSEOUT (ASMBULK EXTID=200 MATOP4=-25)

```

(See also Remarks 17. and 18.)

| Descriptor | Meaning   |
|------------|---|
| STIFFNESS  | Store the boundary stiffness matrix. See Remarks 1. and 2.        |
| MASS       | Store the boundary mass matrix. See Remark 1.                     |
| DAMPING    | Store the boundary viscous damping matrix. See Remarks 1. and 2.  |
| K4DAMP     | Store the boundary structural damping matrix. See Remark 1.       |
| LOADS      | Store the boundary static loads matrix. See Remark 1.             |
| FSCOUP     | Store the boundary fluid-structure coupling matrix. See Remark 1. |

| Descriptor                  | Meaning   |
|-----------------------------|---|
| ASMBULK or<br>ASMBULK = MAN | Generate Bulk Data entries for use in a subsequent superelement assembly process and store them on the assembly punch file (.asm). This data, which is used in the main bulk data portion of a subsequent assembly job, includes an SEBULK entry that specifies MANUAL as the method for searching boundary points, and an SECONCT entry that defines connections for boundary grid and scalar points. See Remarks 3., 6., and 15.  |
| ASMBULK = MANQ              | Similar to the ASMBULK = MAN option, except that the generated SECONCT entry defines connections not only for boundary grid and scalar points, but also for Q-set points. This allows the user to have control over the Q-set points of the external superelement in the subsequent assembly job. See Remarks 3., 5., and 15.   |
| ASMBULK = AUTO              | Generate Bulk Data entries for use in a subsequent superelement assembly process, and store them on the assembly punch file (.asm). This data, which is to be used in the main bulk data portion of a subsequent assembly job, includes an SEBULK entry that specifies AUTO as the method for searching boundary points, and an SECONCT entry that defines connections for boundary scalar points. See Remarks 3., 6., and 15.  |
| EXTBULK                     | Generate Bulk Data entries related to the external superelement and store them on the standard punch file (.pch). This data is used in the BEGIN SUPER portion of the bulk data of a subsequent assembly job. EXTBULK need not be specified if DMIGPCH or MATOP4 (see the following descriptions) is specified. See Remarks 3., 7., 8., and 15.   |
| EXTID= <i>seid</i>          | Note that, in general, the EXTBULK keyword is not required and is provided solely for user convenience. In the absence of EXTBULK and the associated output on the standard punch file resulting from it, the subsequent assembly job will retrieve the required data for the external superelement from the medium on which the boundary matrices are stored.<br><br><i>seid</i> (integer > 0) is the superelement ID to be used on the SEBULK and SECONCT (if applicable) Bulk Data entries stored on the assembly punch file (.asm) if ASMBULK is specified, and in the BEGIN SUPER Bulk Data entry stored on the standard punch file (.pch) if EXTBULK or DMIGPCH is specified. See Remarks 3. - 9., 13., and 17. |
| DMIGSFIX = <i>cccccc</i>    | <i>cccccc</i> is the suffix (up to six characters) to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks 10. - 13. See also Example 3 in Remark 18.   |
| DMIGSFIX = EXTID            | The <i>seid</i> defined by the EXTID keyword is the suffix to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks 3., 10. - 13. See also Example 3 in Remark 18.   |

| Descriptor   | Meaning   |
|--|---|
| MATDB<br>(or MATRIXDB) = dbext<br>(Default)                    | Store the boundary matrices and other information on the database. Dbext is the name of a new dbset on which to store the boundary matrices. Dbext must be allocated on an INIT FMS statement. By default, matrices are stored on DBALL. See Example 1 in Remark 18. Dbext is required if 3-step external superelement data recovery will be performed.   |
| DMIGDB= dbext  | Similar to MATDB (or MATRIXDB) except that the boundary matrices are stored as DMIG Bulk Data entries on the database. Dbext is the name of a new dbset on which to store the boundary matrices. Dbext must be allocated on an INIT FMS statement. By default, matrices are stored on DBALL. See Example 1 in Remark 18. Dbext is required if 3-step external superelement data recovery will be performed.   |
| DMIGOP2 = <i>unitop2</i>                                       | Store the boundary matrices as DMIG Bulk Data entries on an .op2 file whose Fortran unit number is given by <i>unitop2</i> (Integer > 0). See Remark 16. See also Example 2 in Remark 18.   |
| DMIGPCH  | Store the boundary matrices as DMIG Bulk Data entries on the standard punch file (.pch). Note that, if boundary matrices include both structural and fluid grids, use DMIGOP2 instead. See Remarks 3. and 7. - 15. See also Example 3 in Remark 18.   |
| MATOP4 = <i>unitop4</i><br>(or MATRIXOP4 =<br><i>unitop4</i> ) | <p><i>unitop4</i> is a non-zero integer with the following meanings:</p> <p><i>unitop4</i> &gt; 0</p> <p>Store the boundary matrices in <i>sparse</i> format on an .op4 file whose Fortran unit number is given by <i>unitop4</i></p> <p><i>unitop4</i> &lt; 0</p> <p>Store the boundary matrices in <i>non-sparse</i> format on an .op4 file whose Fortran unit number is given by <math> unitop4 </math></p> <p>See Remarks 3., 7., 8., 15., and 16. See also Example 4 in Remark 18.</p>                 |
| ASET/QSET  | Specifies how the external superelement's q-set dofs (dynamic degrees-of-freedom) are defined on the exported file. By default, all q-set dofs are written to QSET/QSET1 entries which means the superelement's modes will be "added" to but not synthesized into the assembly's modes. By specifying ASET, the q-set dofs are written to ASET/ASET1 entries and will be synthesized into the assembly's. The ASET/QSET switch does not affect DMIGPCH media option which always writes ASET/ASET1 entries. |

**Remarks:**

1. If none of the descriptors STIFFNESS through FSCOUP is specified, then all matrices are stored.
2. STIFFNESS and DAMPING may be abbreviated to STIF and DAMP, respectively.

3. EXTID with an *seid* value must be specified if one or more of ASMBULK, EXTBULK, DMIGPCH, or MATOP4 are specified.

If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, the value *seid* may not exceed 999999 since this value becomes part of the names given to the DMIG matrices generated on the standard punch file (.pch). See Remark 13., and Example 3 in Remark 18.

If PARAM,AUTOQSET,YES is specified to used generate the Q-set degrees of freedom (DOFs) (generalized coordinates), the value *seid* may not exceed 999 since this value becomes part of the automatically generated IDs of the SPOINTs representing the Q-set DOFs. See explanation in Item c under Remark 17.

4. If ASMBULK is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK seid ... (specifies MANUAL as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary grid and scalar
points)
```

```
GRID entries for boundary and access grid points
SPOINT entries for boundary and access scalar points
CORD2x entries associated with the boundary and access GRID
entries
```

5. If ASMBULK = MANQ is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK seid ... (specifies MANUAL as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary grid and
scalar points as well as for Q-set points)
GRID entries for boundary and access grid points
SPOINT entries for boundary and access scalar points as well as
for Q-set points
CORD2x entries associated with the boundary and access GRID
entries
```

6. If ASMBULK = AUTO is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK seid ... (specifies AUTO as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary scalar
points)
```

```
GRID entries for access grid points
CORD2x entries associated with the access grid points
SPOINT entries for boundary and access scalar points
```

7. If DMIGPCH or MATOP4 is specified, then EXTBULK need not be specified. Also, the model cannot have rotors (ROTOR, ROTORG, ROTORSE, and ROTORAX Bulk Data entries).
8. If DMIGPCH is not specified, but EXTBULK or MATOP4 is specified, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

```
BEGIN BULK seid
GRID entries for boundary points
GRID entries for interior points referenced by PLOTEL entries
SPOINT entries for boundary scalar points as well as for Q-set
points
CORD2x entries associated with the above GRID entries
EXTRN
ASET
QSET/QSET1
PLOTEL
```

9. If DMIGPCH is specified, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

```
BEGIN SUPER seid
GRID entries for boundary points
SPOINT entries for boundary scalar points as well as for Q-set
points
CORD2x entries associated with the boundary GRID entries
ASET/ASET1
PLOTEL entries referencing the boundary GRID entries
DMIG entries for the requested boundary matrices
```

10. The DMIGSFIX keyword is ignored if DMIGPCH is not specified.
11. If DMIGPCH is specified without the DMIGSFIX keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

```
KAAx (boundary stiffness matrix)
MAAx (boundary mass matrix)
BAAx (boundary viscous damping matrix)
K4AAx (boundary structural damping matrix)
PAX (boundary load matrix)
AAX (boundary fluid-structure coupling matrix)
```

See Example 3 in Remark 18.

12. If the DMIGSFIX = *cccccc* form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

```
Kcccccc (boundary stiffness matrix)
Mcccccc (boundary mass matrix)
Bcccccc (boundary viscous damping matrix)
K4cccccc (boundary structural damping matrix)
Pcccccc (boundary load matrix)
Acccccc (boundary fluid-structure coupling matrix)
```

See Example 3 in Remark 18.

13. If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

```
Kseid (boundary stiffness matrix)
Mseid (boundary mass matrix)
Bseid (boundary viscous damping matrix)
K4seid (boundary structural damping matrix)
```

`Pseid` (boundary load matrix)  
`Aseid` (boundary fluid-structure coupling matrix)  
 where `seid` is the superelement ID specified by the `EXTID` keyword.  
 See Example 3 in Remark 18.

14. If the `DMIGPCH` option is specified, the boundary DMIG matrices generated and stored on the standard punch file (.pch) may not be as accurate as the boundary matrices resulting from the other options (`MATDB/MATRIXDB`, or `DMIGOP2` ,or `MATOP4`). Accordingly, this may result in decreased accuracy from the subsequent assembly job using these DMIG matrices.
15. The punch output resulting from `EXTSEOUT` usage is determined by `ASMBULK`, `EXTBULK`, `DMIGPCH`, and `MATOP4` as follows:
  - No `ASMBULK`, `EXTBULK`, `DMIGPCH` or `MATOP4`  
 No punch output.
  - `ASMBULK`, but no `EXTBULK`, or `DMIGPCH`, or `MATOP4`  
 Punch output is generated and stored on the assembly punch file (.asm) as indicated in Remarks 4. through 6.
  - No `ASMBULK`, but `EXTBULK`, `DMIGPCH`, or `MATOP4`  
 Punch output is generated and stored on the standard punch file (.pch) as indicated in Remarks 8. or 9. (as appropriate).
  - `ASMBULK` and `EXTBULK`, `DMIGPCH` or `MATOP4`  
 Punch output consists of two distinct and separate parts. One part is generated and stored on the assembly punch file (.asm) as indicated in Remarks 4. through 6. The other part is generated and stored on the standard punch file (.pch) as indicated in Remarks 8. or 9. (as appropriate).
16. If `DMIGOP2 = unitop2` or `MATOP4 = unitop4` is specified, an appropriate `ASSIGN OUTPUT2` or `ASSIGN OUTPUT4` statement must be present in the File Management Section (FMS) for the specified Fortran unit numbers. See Examples 2 and 4 in Remark 18.
17. The creation of an external superelement using `EXTSEOUT` involves running a Nastran job, with the following additional data:
  - a. The data for the creation of the external superelement is specified by the `EXTSEOUT` Case Control command. It is best to specify this command above the subcase level. However, if the Nastran job itself involves superelements, then the `EXTSEOUT` command may also be specified within the subcase that is applicable to the residual.
  - b. The boundary points of the external superelement are specified by `ASET/ASET1` Bulk Data entries.
  - c. If the external superelement creation involves component mode reduction, then Q-set degrees of freedom (DOFs) (generalized coordinates) must be defined. This can be done either by automatically defining them via `PARAM,AUTOQSET,YES` or by explicitly specifying them via `QSET/QSET1` Bulk Data entries. Note that for the latter case of explicit definition, only `SPOINTs` may be specified as Q-set points; no grid points may be specified. If this requirement is not met, the program terminates the execution with an appropriate fatal message.

If PARAM,AUTOQSET,YES is specified, SPOINTs are automatically generated internally to represent the Q-set DOFs. The IDs of these SPOINTs are of the form 9 $sssnnnn$  where  $sss$  is the superelement ID  $seid$  specified by the EXTID keyword, and  $nnnn$  is a mode number. Both  $sss$  and  $nnnn$  will have leading zeros inserted in them to ensure that  $sss$  is a three-digit number and  $nnnn$  is a four-digit number. Thus, for example, the Q-set DOF corresponding to the 8<sup>th</sup> mode of superelement ID 5 would be represented by an SPOINT with an automatically generated ID of 90050008, while the Q-set DOF corresponding to the 50<sup>th</sup> mode of superelement ID 25 would be represented by an SPOINT with an automatically generated ID of 90250050.

Because of the preceding numbering scheme, the superelement ID  $seid$  specified by the EXTID keyword must necessarily not exceed 999 when PARAM,AUTOQSET,YES is specified. The program terminates the job with a User Fatal Message if this condition is not met.

Further, because of the preceding numbering scheme, the user should ensure that, when PARAM,AUTOQSET,YES is specified, the IDs of the grid and scalar points of the external superelement do not conflict with the automatically generated SPOINT IDs of the Q-set DOFs. The program terminates the job with a User Fatal Message if this condition is not met.

- d. The fixity of the boundary DOFs for the component mode reduction may be specified using the BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 Bulk Data entries. (The default scenario assumes that all boundary DOFs are fixed for component mode reduction.)
- e. The output for the external superelement is generated in the assembly job. This output consists of displacements, velocities, accelerations, SPC forces, MPC forces and element stresses, strains and forces. However, in order for this output to be generated in the assembly job, the output requests must be specified in the external superelement creation run. Normally, the only output requests for the external superelement that are honored in the assembly job are those that are specified in the creation run. There is, however, one important exception to this: the output for the boundary and access grid and scalar points, as well as for all grid points associated with PLOTEL entries, can be obtained in the assembly job *even if there is no output request specified for these points in the creation run*.

If the creation run contains the load requests TEMPERATURE (LOAD) or DEFORM, then element stresses and element forces will not be available in the assembly run.

- f. If the assembly job involves the use of PARAM Bulk Data entries, then the following points should be noted:
  - PARAM entries specified in the main bulk data portion of the input data apply *only to the residual and not to the external superelements*.
  - PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for an external superelement apply *only to that superelement*.
  - The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all external superelements, is to specify such PARAM entries in the Case Control Section, not in the main bulk data. This is particularly relevant for PARAMs such as POST.

18. The following examples illustrate details of job setups for the external superelement creation and the subsequent assembly process for various scenarios. These examples assume that there are three external superelement creation jobs, one each for external SE 10 (extse10.dat), SE 20 (extse20.dat) and SE 30 (extse30.dat), followed by an assembly job.

### Example 1. MATDB / MATRIXDB or DMIGDB Option

- External SE Creation Jobs

- File Management Section FMS Requirement only if 3rd step external superelement data recovery is desired

SE 10: INIT EXTDB

SE 20: INIT EXTDB

SE 30: INIT EXTDB

These INIT statements along with the DBEXT keyword below will create separate DBsets called “extse10.EXDTB”, “extse20.EXDTB”, and “extse30.EXDTB” and store the necessary data for the Assembly Job on them. All other data will be stored on DBALL.

- Case Control Requirement for the MATDB / MATRIXDB option if 3<sup>rd</sup> step external superelement data recovery is **not** desired.

SE 10: EXTSEOUT (ASMBULK EXTID = 10)

SE 20: EXTSEOUT (ASMBULK EXTID = 20)

SE 30: EXTSEOUT (ASMBULK EXTID = 30)

The EXTBULK keyword may be specified, but it is not necessary.

- Case Control Requirement for the DMIGDB option if 3<sup>rd</sup> step external superelement data recovery is **not** desired.

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGDB)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGDB)

For both options, scr = no should be specified on the Nastran job command line to ensure that the databases are saved at the end of the jobs.

- Case Control Requirement for the MATDB / MATRIXDB option if 3<sup>rd</sup> step external superelement data recovery is desired.

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DBEXT=SE10)

SE 20: EXTSEOUT (ASMBULK EXTID = 10 DBEXT=SE20)

SE 30: EXTSEOUT (ASMBULK EXTID = 10 DBEXT=SE30)

- Case Control Requirement for the DMIGDB option if 3<sup>rd</sup> step external superelement data recovery is desired.

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB DBEXT=SE10)

**Example 1. MATDB / MATRIXDB or DMIGDB Option (continued)**

SE 20: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB DBEXT=SE20)

SE 30: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB DBEXT=SE30)

## ■ Assembly Job

- File Management Section (FMS) Requirement

(Note: If DBEXT was specified on the EXTSEOUT command in the External SE Creation Jobs then the extse10.DBALL, etc. files may be hidden (moved or renamed) from the Assembly Job for purposes of security or privacy.)

ASSIGN dbname10='extse10.MASTER'

ASSIGN dbname20='extse20.MASTER'

ASSIGN dbname30='extse30.MASTER'

DBLOCATE DATABLK=(EXTDB, EXTROTDB) CONVERT(SEID=10)

LOGICAL=dbname10

DBLOCATE DATABLK=(EXTDB,EXTROTDB) CONVERT(SEID=20)

LOGICAL=dbname20

DBLOCATE DATABLK=(EXTDB, EXTROTDB) CONVERT(SEID=30)

LOGICAL=dbname30

(Note: All of the data blocks stored on the databases for the external SEs have the same common name of EXTDB or EXTROTDB for external SEs with rotors.)

If 3rd step data recovery is desired for the external superelement then see the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the Main Bulk Data.

INCLUDE 'extse10.asm'

INCLUDE 'extse20.asm'

INCLUDE 'extse30.asm'

**Example 2. DMIGOP2 Option**

## ■ External SE Creation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2='extse10\_op2' UNIT=25 DELETE

ASSIGN OUTPUT2='extse20\_op2' UNIT=26 DELETE

ASSIGN OUTPUT2='extse30\_op2' UNIT=27 DELETE

### Example 2. DMIGOP2 Option (continued)

- Case Control Requirement

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGOP2 = 25)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGOP2 = 26)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGOP2 = 27)

The EXTBULK keyword may be specified, but it is not necessary.

scr = yes may be specified on the Nastran command line unless a 3rd step external superelement data recovery restart is desired.

- Assembly Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2='extse10\_op2' UNIT=25

ASSIGN INPUTT2='extse20\_op2' UNIT=26

ASSIGN INPUTT2='extse30\_op2' UNIT=27

If 3rd step data recovery is desired for the external superelement then see the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

INCLUDE 'extse10.asm'

INCLUDE 'extse20.asm'

INCLUDE 'extse30.asm'

### Example 3. DMIGPCH Option

- External SE Creation Jobs

- Case Control Requirement

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGPCH)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGPCH,  
DMIGSFIX = XSE20)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGPCH,  
DMIGSFIX = EXTID)

scr = yes may be specified on the Nastran command line since there is no need for the databases to be saved at the end of the jobs.

- Assembly Job

- Case Control Requirement

**Example 3. DMIGPCH Option (continued)**

K2GG = (KAAx, KXSE20, K30)

M2GG = (MAAx, MXSE20, M30)

B2GG = (BAAx, BXSE20, B30)

K42GG = (K4AAx, K4XSE20, K430)

P2G = (PAX, PXSE20, P30)

A2GG = (AAX, AXSE20, A30)

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

INCLUDE 'extse10.asm'

INCLUDE 'extse20.asm'

INCLUDE 'extse30.asm'

The following INCLUDEs are also required. They must be grouped together and specified at the very end of the main bulk data (just before the ENDDATA delimiter).

INCLUDE 'extse10.pch'

INCLUDE 'extse20.pch'

INCLUDE 'extse30.pch'

**Example 4. MATOP4 Option**

- External SE Creation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT4='extse10\_op4' UNIT=25 DELETE

ASSIGN OUTPUT4='extse20\_op4' UNIT=26 DELETE

ASSIGN OUTPUT4='extse30\_op4' UNIT=27 DELETE

- Case Control Requirement

SE 10: EXTSEOUT (ASMBULK EXTID = 10 MATOP4 = 25)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 MATOP4 = 26)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 MATOP4 = -27)

Note: Boundary matrices will be stored in *sparse* format on .op4 files with Fortran unit numbers 25 and 26 and in *non-sparse* format on the .op4 file with Fortran unit number 27.

scr = yes may be specified on the Nastran command line unless a 3rd step external superelement data recovery restart is desired.

- Assembly Job

#### Example 4. MATOP4 Option (continued)

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT4='extse10_op4' UNIT=25
```

```
ASSIGN INPUTT4='extse20_op4' UNIT=26
```

```
ASSIGN INPUTT4='extse30_op4' UNIT=27
```

If 3rd step data recovery is desired for the external superelement then see the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

```
INCLUDE 'extse10.asm'
```

```
INCLUDE 'extse20.asm'
```

```
INCLUDE 'extse30.asm'
```

The following INCLUDEs are also required. They must be grouped together and specified at the very end of the main bulk data (just before the ENDDATA delimiter).

```
INCLUDE 'extse10.pch'
```

```
INCLUDE 'extse20.pch'
```

```
INCLUDE 'extse30.pch'
```

19. If EXTBULK is specified, then METADATA entries will be copied to the .pch file.
20. Data recovery for the external superelement is possible via two methods: two-step and three-step.
  - a. In the two-step method, EXTSEOUT automatically generates output transformation matrices (OTMs) based on STRESS, FORCE, DISPLACEMENT, SPCFORCE, MPCFORCE, and STRAIN Case Control commands specified in the first (creation) run. Then in the second (assembly) run, data recovery is performed on the external superelement. If Modules are present in the first run, then OTMs are generated for those grid points and elements in Module 0 only.
  - b. The three-step method requires a restart of the first run to perform data recovery on the external superelement. The method is explained in the description of the EXTDRIN Case Control command. If Modules are present in the first run, this method allows for data recovery in any Module.
  - c. Both methods are described in more detail in the *MSC Nastran Superelements and Modules User's Guide*.

- d. For dynamically reduced external superelements, the displacement OTMs created with dynamic reduction include both the dynamic effects of the o-set and also the fixed boundary effects of the o-set due to interior loads. With the two-step method, only one of these effects should be included in a SOL 101, 106 and 400 (with ANALYSIS=STATICS) assembly. Therefore, if there are interior loads in the external superelement and the two-step method is being used, attaching the dynamically reduced external superelements in a static analysis of the assembly is not recommended.

**FATIGUE****Fatigue Output Request**

Requests one or more fatigue analyses for use in pseudo-static (SOL 101), modal (SOL 103), modal transient (SOL 112), and frequency response (SOL 108 and SOL 111) runs.

Format (SOL 101, 103, 112, 200 with ANALYSIS=STATICS):

$$FATIGUE \left[ \left( \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, FORMAT = [CODE], \begin{bmatrix} BULK \\ SET \end{bmatrix}, \begin{bmatrix} STATICS \\ \end{bmatrix}, STROUT = [CODE] \right) \right] = \left\{ n \right\}$$

Format (SOL 108, 111, 200 with ANALYSIS=DFREQ or MFREQ) (See Remark 12.):

$$FATIGUE \left[ \left( \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, FORMAT = [CODE], \begin{bmatrix} BULK \\ \end{bmatrix}, \begin{bmatrix} FREQ \\ DFREQ \\ MFREQ \end{bmatrix} \right) \right] = \left\{ n \right\}$$

Examples:

FATIGUE=100

SET 99 = 100, 200

FATIGUE (SET)=99

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Printed output will be presented as a tabular listing of elements for each fatigue analysis or event. In the case of duty cycle, where multiple events are output (EVNTOUT=1 on FTGSEQ bulk data), each event is presented as a separate fatigue analysis. Output written to the OEFTG data blocks is always in SORT1 format.                           |
| SORT2      | Printed output will be presented as a tabular listing of events for each element, node, or angle. This is only applicable for duty cycle (loading sequences with more than one event - EVNTOUT=1 on FTGSEQ bulk data). If there is only one event, the output is identical to SORT1. Output written to the OEFTG data blocks is always in SORT1 format. |

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

See Remark 15. regarding data blocks produced by the FATIGUE case control.

|          |  |
|----------|--|
| FORMAT   | Request that fatigue be output in specific file formats. See Remark 6. and 11.   |
| STROUT   | Request that certain stress or strain output be included. See Remarks 13. and 14.  |
| CODE     | Codes for specific file format or stress output. See Remark 6., 11., and 13.   |
| BULK     | Specifies that the <i>n</i> refers directly to bulk data IDs of FTGSEQ/ FTGLOAD, FTGPARM and FTGDEF entries of the same ID (default) to process.   |
| SET      | Specifies that the <i>n</i> refers to a previously appearing SET ID (SOL 101, 103, & 112 only) containing the IDs of FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries to process. This is a mechanism to request multiple fatigue output requests in a single analysis run. |
| STATICS  | <b>SOL 200 only.</b> Specifies the FATIGUE case control is for ANALYSIS=STATICS subcases in a SOL 200 optimization run. This is the default if not specified. See Remark 9.  |
| FREQ     | <b>SOL 200 only.</b> Either of these specifies the FATIGUE case control is for frequency response (ANALYSIS=MFREQ or DFREQ) subcases in a SOL 200 optimization run.  |
| DFREQ    | This is not the default and must be specified if the FATIGUE case control is referenced on a DRESP1 of type FRFTG. See Remark 9.   |
| MFREQ    |  |
| <i>n</i> | ID of FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries of the same ID (BULK option) to process, or  |
|          | ID of a previously occurring SET case control entry (SET option) containing the IDs of the FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries to process (SOL 101, 103, & 112 only).  |

#### Remarks:

1. A single FATIGUE case control is required to perform one (SOL 108/111) or more (SOL 101/103/112) fatigue analyses and must be present above the SUBCASE level. If not present, no fatigue analysis will occur regardless of the presence of other bulk data related to fatigue analysis.

2. It is not necessary to include any case control STRESS and/or STRAIN output requests in order for the FATIGUE output request to obtain the necessary stresses or strains for the fatigue calculation.
3. For a single fatigue analysis, BULK=n points to a FTGDEF, a FTGPARM, and a FTGSEQ (or FTGLOAD) entry, each with ID=n.
4. For multiple fatigue analyses, SET=n points to a previously appearing SET case control and each member of the SET is the ID of a FTGDEF, a FTGPARM, and a FTGSEQ (or FTGLOAD) bulk data entry with that ID (**SOL 101, 103, or 112 only**).
5. A fatigue analysis must have, at a minimum, loading and material data defined on either FTGSEQ or FTGLOAD and MATFTG bulk data entries, respectively, for a fatigue analysis to be valid. FTGPARM and FTGDEF entries can be absent, in which case, defaults will be used.
6. The following additional, optional, fatigue output file formats may be requested. The given codes must be summed if multiple files are requested. Example: CSV file and FEF file CODE would be  $64+128=192$ . Note the following:
  - **SOL 101, 103, or 112:** For multi-layered results, such as shells that compute top and bottom results, a separate file for each is generated for HYP, UNV, and FEF formats with \_top or \_bottom designations in the name. If only a single file with worst case layer results is desired, add 256 or 10,000 to the CODE. Using the above example would give 448 or 10192. Either mechanism works. Note that CSV and FER file output only ever creates a single file as both layers are reported in the single file, but adding 256 or 10,000 writes only the worst case layer result.
  - **SOL 108, or 111:** For multi-layered results, such as shells that compute top and bottom results, a separate file for each is generated for FEF formats with the following designations in the name: \_N or \_E (for element nodal or element center, respectively), \_W or \_T or \_B (for worst layer, top layer or bottom layer, respectively) or \_EVi (for event i, or \_EVS for summary only).
  - Some output is not available for duty cycle jobs if EVNTOUT>0 on the FTGSEQ bulk data entry. Check messages if the f06 and log files if the requested output files are not being generated.

| File Format                              | CODE for SOLs 101/103/112 | CODE for SOLs 108/111 |
|--|---------------------------|-----------------------|
| No additional output                     | 0                         | 0                     |
| FER (Design Life Output File)            | 4                         | -                     |
| HYP (HyperMesh Results File)             | 8                         | -                     |
| UNV (Universal File)                     | 32                        | -                     |
| CSV File (Comma Separated - Excel File)* | 64                        | 64                    |
| FEF (Patran Results File)†               | 128                       | 128                   |

\* For random vibration fatigue using SOLs 108 or 111, the CSV file output request is also dependent on the LOGLVL field setting on the FTGPARM entry. By default a <jid>PSD.csv file is always generated, which at a minimum contains the input PSD, stress transfer function (TF) and result PSD for the critical entity.

†SOL 101, 103, 112 produce FEF files of type 'PATRAN 2 els' or 'PATRAN 2 nod' with .fef extension; SOL 108, 111 produce FEF files of type 'PATRAN 2 els' with .els\_fef extension. Also see Remark 11.

7. RESTARTs are possible with FATIGUE output request as long as the stress/strain state of the entities requested does not change. Most parameters on the FTGPARM (LOC excluded as it requires a different set of stresses/strains) can be changed on a RESTART. Different loading (FTGSEQ) and materials (MATFTG) can also be investigated using the RESTART capability (SOLs 101, 103, & 112 only).
8. Standard fatigue output is:

| Stress-Life (SN) Analysis | Strain-Life (eN) Analysis |
|---------------------------|---------------------------|
| LIFE (Repeats*)           | LIFE (Repeats)            |
| LOG of LIFE (Repeats)     | LOG of LIFE (Repeats)     |
| LIFE (user units)†        | LIFE (user units)         |
| LOG of LIFE (user units)  | LOG of LIFE (user units)  |
| DAMAGE                    | DAMAGE                    |
| LOG of DAMAGE             | LOG of DAMAGE             |
| MAX STRESS‡               | MAX STRESS or STRAIN **   |
| MIN STRESS                | MIN STRESS or STRAIN      |

\*Or Seconds for SOL 108 and SOL 111

†User units are fatigue equivalent units as defined on the FTGSEQ or FTGLOAD entry and other output is available depending on the settings of the FTGPARM entry. Please see those entries for more detail.

‡For SOLs 108 and 111 Maximum Stress = Mean Stress + MAXPEAK\*RMS Stress, Minimum Stress = Mean Stress - MAXPEAK\*RMS Stress. MAXPEAK is defined on FTGPARM entry and is defaulted to 3.0.

\*\*Depends on FTGPARM line used: STRESS or STRAIN.

9. STATICS, FREQ, DFREQ, MFREQ options are utilized in SOL 200 only. Fatigue and design sensitivity analysis will be performed for the designated analysis type only. If STATICS is specified (default), FATIGUE case control must be associated with subcases containing ANALYSIS=STATIC for a pseudo-static fatigue analysis and the fatigue DRESP1 responses/constraints specified must be of RTYPE=FATIGUE. If one of FREQ, DFREQ, MFREQ is specified, FATIGUE case control must be associated with subcases containing ANALYSIS=MFREQ or DFREQ for random vibration fatigue analysis and the fatigue DRESP1 response/constraints specified must be of RTYPE=FRFTG.
10. For SOL 108/111 with FATIGUE analysis, PARAM,SPARSED, no and PARAM,DDRMM,-1 both must be present. Otherwise, FATIGUE analysis will be skipped. Or, PARAM,SPARSED, yes, which is default, can be used instead.

11. The FEF file output request produces PATRAN 2 style elemental or nodal ASCII results files that can be imported directly into Patran for post processing. In order to import these files into Patran a template file is also required in order to tell Patran how to map the columns of data in the files to proper, meaningful labels. The analysis job produces both the FEF file and the corresponding template files. For SOLs 101, 103, and 112, the template file has the extension .tmp1 with the same base name as the FEF file. For SOLs 108 and 111, the template file is called <jid>.res\_tmpl1, where <jid> is the job name. Jobs run with LOC=ELEM or NODE on the FTGPARM entry create elemental based FEF files or all SOL sequences. Jobs run with LOC=NODA produce nodal based FEF files for SOLs 101, 103, and 112 and produce elemental based FEF files for SOLs 108 and 111. In Patran these files are imported under the **File|Import** menu with the **Object** set to **Results** and the **Format** set to either **PATRAN 2 els...** or **PATRAN 2 nod...** for elemental or nodal based FEF results files, respectively. The **Element Node Results** switch must be set to **Connectivity Based** for proper import of elemental based results.

You may use the job produced template files to import the FEF files or you may use standard template files provided with the Patran installation. The advantage of using the standard template files is that you may select them directly from the list without navigating the file browser to the location of the job produced template files. The disadvantage is that these files use more generic labels. The job produced template files use labels that indicate the stress units and actual defined equivalent fatigue units as defined by the job. When using the standard template files you must know which template file to select based on the table below.

| Job Type (SOLs 101, 103, 112)                | Template Base File Name<br>(extension .res_tmpl1) |
|--|---|
| Standard SN Analysis                         | NEF_standard_SN_fef                               |
| Standard εN Analysis                         | NEF_standard_EN_fef                               |
| Standard Spot Weld Analysis                  | NEF_standard_SPOT_fef                             |
| Standard Seam Weld Analysis                  | NEF_standard_SEAM_fef                             |
| Standard SN Factor of Safety Analysis        | NEF_standard_SN_FOS_fef                           |
| Standard εN Factor of Safety Analysis        | NEF_standard_EN_FOS_fef                           |
| Standard Spot Weld Factor of Safety Analysis | NEF_standard_SPOT_FOS_fef                         |
| Standard Seam Weld Factof of Safety Analysis | NEF_standard_SEAM_FOS_fef                         |
| SN + Multiaxial METHOD=SIMPLE                | NEF_multSimpl_SN_fef                              |
| SN + Multiaxial METHOD=STANDARD              | NEF_multStnd_SN_fef                               |
| SN + Multiaxial METHOD=AUTO                  | NEF_multAuto_SN_fef                               |
| εN + Multiaxial METHOD=SIMPLE                | NEF_multSimpl_EN_fef                              |
| εN + Multiaxial METHOD=STANDARD              | NEF_multStnd_EN_fef                               |
| εN + Multiaxial METHOD=AUTO                  | NEF_multAuto_EN_fef                               |
| Seam Weld + Multiaxial METHOD=SIMPLE         | NEF_multSimpl_SEAM_fef                            |

| Job Type (SOLs 101, 103, 112)                 | Template Base File Name<br>(extension .res_tmpl) |
|---|--|
| Seam Weld + Multiaxial METHOD=STANDARD        | NEF_multStnd_SEAM_fef                            |
| SN + FOS + Multiaxial METHOD=SIMPLE           | NEF_multSimp_SN_FOS_fef                          |
| $\epsilon$ N + FOS + Multiaxial METHOD=SIMPLE | NEF_multSimp_EN_FOS_fef                          |
| Seam Weld + FOS + Multiaxial METHOD=SIMPLE    | NEF_multSimp_SEAM_FOS_fef                        |
| Job Type (SOLs 108, 111)                      | Template Base File Name<br>(extension .res_tmpl) |
| Standard SN or $\epsilon$ N Analysis          | NEFV_standard_fef                                |

12. When a FATIGUE case control is present to perform random vibration fatigue analysis using SOLs 108 and 111, only one FREQUENCY case control set is allowed for all transfer function SUBCASEs and should be above the SUBCASE level or only present in 1st SUBCASE. The FREQUENCY set cannot change from SUBCASE to SUBCASE or a fatal error is issued.
13. SOL 101, 103, or 112 only: The following additional, optional, stress or stain output may be requested. The given codes must be summed if multiple requests are required. Example: CODE would be 1+2=3 for fatigue stress and maximum stress range vector results. The presence of this output request overrides the STROUT entry on the FTGPARM entry. If PLOT is specified in the FATIGUE case control, no output is printed and only available in the OUTPUT2 or HDF5 files.

| Additional Stress/Strain Output   | CODE |
|---|------|
| No additional output  | 0    |
| <b>Fatigue Stress/Strain</b> - these are the physical (or modal) stress/strain tensors passed to the fatigue analysis for each requested entity of the fatigue analysis for each SUBCASE (or mode). Valid for SOL 101, 103, and 112 only.   | 1    |
| <b>Maximum Stress/Strain Range Magnitude or Vector</b> - this is the stress/strain maximum range magnitude (for all scalar COMB values of the FTGPARM entry) or vector (for 2D critical plane* analysis, COMB=CRITICAL) for all requested entities at the critical angle for SOLs 101, 103, and 112 only, as returned from the fatigue analysis. When LOC=ELEM or NODE on the FTGPARM entry, the computed vector is in the basic coordinate system. For LOC=NODA, the computed vector is in the SURFACE system as defined by OUTPUT field on the NAVG line of the FTGPARM entry. For scalar COMB values, a vector is not computed and only the x-component of the vector is provided. The other two components are always zero in this case.                                    | 2    |
| <b>Stress/Strain Scalar Response Time History</b> - this is the computed scalar response time history (for all COMBs on the FTGPARM entry) at all time points as returned from the fatigue analysis for SOLs 101, 103, and 112 only. This is the actual response from which rainflow ranges and damage are computed. While individual events are processed, the ALL event is not processed when METHOD=0 on FTGPARM. Also LAYER=1 or 2 on the FTGPARM entry is honored for STROUT=4, but LAYER=0 prints values for both top and bottom. To limit output and ensure decent performance maxENTS on the FTGDEF entry is defaulted to 100 and if the number of entities exceeds this, the job stops. Setting maxENTS to a high number is not recommended due to performance issues. | 4    |
| <b>Stress/Strain Tensor Time History</b> - this is the computed tensor time history response at all time points for all requested entities used by the fatigue analysis for SOL 112 only.   | 8    |

\* COMB = CRITICAL on the FTGPARM entry.

14. STROUT is not honored in manual restarts using SOL FTGRSTR (automatic restarts are OK). STROUT=4 may use additional memory space on your system. For STROUT=4, it is also highly recommended to limit the number of requested entities on the FTGDEF bulk data entry. If the job stops because too many entities are requested, you can set maxENTS on the FTGDEF entry. STROUT=4 has the potential to produce enormous amounts of response data and external files, which can severely affect performance and disk space. STROUT=4 produces temporary CSV response files of each entity. These CSV files are deleted after the job completes unless scr=no is used when submitting the job. When METHOD=0 on the FTGPARM entry, STROUT=4 only produces responses for each event. Only when METHOD=1 or 2 will STROUT=4 produce responses for all events combined or if there is only one event.

15. Output Data Blocks: The following table indicates the output data blocks produced by the FATIGUE case control and which parameters trigger creation of those data blocks. See the DMAP Programmers Guide for a description of these data blocks.

| Data Block | Comment   |
|------------|---|
| OEFTG      | <b>SOL 101, 103, 112 only.</b> This is the standard output data block for that is produced simply by the presence of a FATIGUE case control containing fatigue life/damage and related data for time-based fatigue analysis. Data from STROUT=2 is also written to this data block. |
| OEFTGM     | <b>SOL 101, 103, 112 only.</b> This is the multiaxial/biaxial output data block that is produced by the presence of a FATIGUE case control with the usage of the MULTI key word on the FTGPARM entry for time-based fatigue analysis  |
| OEFTGV     | <b>SOL 108, 111 only.</b> This is the standard output data block for that is produced simply by the presence of a FATIGUE case control containing fatigue life/damage and related data for frequency-based fatigue analysis.  |
| OES1FS     | <b>SOL 101 only.</b> This is a physical stress tensor data block like OES that is created due to the presence of STROUT=1 usage in a FATIGUE case control.  |
| OESFTGR    | <b>SOL 103, 112 only.</b> This is a modal stress tensor data block like OES that is created due to the presence of STROUT=1 usage in a FATIGUE case control.  |
| OEFTGRS    | <b>SOL 101, 103, 112 only.</b> This is the scalar stress history response data block created due to the presence of STROUT=4 usage in a FATIGUE case control.   |
| OES1FS     | <b>SOL 112 only.</b> This is a tensor stress history data block like OES that is created due to the presence of STROUT=8 usage in a FATIGUE case control.   |

**FBODYLD****Free Body Load Output Request**

Selects a set of submodels for which free body loads are to be produced and stored.

Format:

$$FBODYLD(LID) = \left\{ \begin{array}{c} ALL \\ name_1, name_2, name_3, \dots \end{array} \right\}$$

Examples:

FBODYLD=ALL

FOBODYLD(100)=WINGLD

FBODYLD(200)=WINGLD,TAILLD

| Descriptor | Meaning   |
|------------|---|
| LID        | Optional user-defined load ID. If LID is not supplied, the subcase ID is used to define this value. |
| ALL        | Loads will be produced for all FBODYLD Bulk Data entries.   |
| name $i$   | Name of an FBODYLD Bulk Data entry that defines the submodel to be used for the load.               |

Remarks:

1. It is recommended, but not required, that the LID be unique across subcases.
2. A separate load is created for each name $i$ .
3. The name list supplies one or more names separated by comma or blank.
4. Each load is stored individually as a one column matrix that is qualified by LID, name $i$ , submodel name, loadcase label, and submodel label (where submodel name is the name on the FBODYSB Bulk Data entry, loadcase label is the label on the FBODYLD Bulk Data entry, and submodel is the label on the FBODYSB Bulk Data entry).

**FEMCHECK**

Specifies model checking options

Specifies model checking options at the start of the run. Specifically checks for RBE3 elements with unconnected independent (Gij) grids and RBE2 elements with unconnected dependent (GMi) grids. Also validates some other case controls in Frequency and Transient analysis before start of the run. RBE2 will only result in WARNING messages, and all other options will result in FATAL messages.

**Format:**

```
FEMCHECK = {NONE}
           {ALL}
           {RBE3, RBE2, DLOAD, FREQ, SDAMP, TSTEP}
```

| Descriptor | Meaning   |
|------------|---|
| NONE       | No checking. This is the default.   |
| ALL        | Turns on all checking listed in the following.  |
| RBE3       | To make sure every independent (Gij) grid on a RBE3 bulk data entry is attached to an element, PLOTEL, or DMIG.   |
| RBE2       | To give a WARNING message for each dependent (GMi) grid on a RBE2 bulk data entry that is not attached to an element, PLOTEL, or DMIG.  |
| DLOAD      | For frequency analysis, check if DLOAD case control is specified. For transient analysis, check if DLOAD or IC case control is specified. Make sure DLOAD case control refers to a valid bulk data entry, i.e. DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, ACLOAD. |
| FREQ       | Check if FREQUENCY case control is specified for frequency analysis. FREQ case control must also refer to a valid bulk data entry, i.e. FREQ, FREQ1, FREQ2, FREQ3, FREQ4, FREQ5.  |
| SDAMP      | Check if SDAMPING Case Control references a valid Bulk Data entry. It must refer to a valid bulk data entry, i.e. TABLED1, TABLED2, TABLED3, TABLED4, TABLED5, TABDMP1.   |
| TSTEP      | For SOL 108/109/111/112, TSTEP case control must refer to a TSTEP bulk data entry.<br>For SOL 129/159, TSTEP case control should refer to a TSTEPNL bulk data entry.  |

**Remark:**

1. This command should be applied above all SUBCASEs.

**FLSFSEL**

## Control for Fluid-Structure Frequency Selection

Control for fluid-structure frequency selection.

Format:

$$\begin{aligned} \text{FLSFSEL} & \left[ \text{LFREQFL} = \left\{ \begin{matrix} 0.0 \\ fl_1 \end{matrix} \right\} \right], \left[ \text{HFREQFL} = \left\{ \begin{matrix} 1. + 30 \\ fl_2 \end{matrix} \right\} \right], \\ & \left[ \text{LFREQ} = \left\{ \begin{matrix} 0.0 \\ fs_1 \end{matrix} \right\} \right], \left[ \text{HFREQ} = \left\{ \begin{matrix} 1. + 30 \\ fs_2 \end{matrix} \right\} \right], \\ & \left[ \text{LMODESFL} = \left\{ \begin{matrix} 0 \\ mf \end{matrix} \right\} \right], \left[ \text{LMODES} = \left\{ \begin{matrix} 0 \\ ms \end{matrix} \right\} \right], \\ & \left[ \text{FLUIDSE} = \left\{ \begin{matrix} 0 \\ seidf \end{matrix} \right\} \right] \end{aligned}$$

Example:

FLSFSEL      HFREQ = 4 .      HFREQFL = 9

| Descriptor | Meaning  |
|------------|--|
| LFREQFL    | Requests in Hertz, lower bound frequency for modal fluid calculations.                             |
| $fl_1$     | Lower frequency range for fluid, real number.  |
| HFREQFL    | Requests in Hertz, upper bound frequency for modal fluid calculations.                             |
| $fl_2$     | Upper frequency range for fluid, real number.  |
| LFREQ      | Requests in Hertz, lower bound frequency for modal structure calculations.                         |
| $fs_1$     | Lower frequency range for structure, real number.  |
| HFREQ      | Requests in Hertz, upper bound frequency for modal structure calculations.                         |
| $fs_2$     | Upper frequency range for structure, real number   |
| LMODESFL   | Lowest modes for fluid portion of model, 0 implies LFREQFL-HFREQFL will determine number of modes. |
| mf         | Number of lowest modes to use for fluid portion of model.  |
| LMODES     | Lowest modes for structure portion of model, 0 implies LFREQ-HFREQ will determine number of modes. |
| ms         | Number of lowest modes to use for structure portion of model.                                      |
| FLUIDSE    | Defines a specified superelement to be used for fluids only.                                       |
| seidf      | Defines a fluid only superelement.   |

**Remarks:**

1. This entry represents a collection of PARAM,name,value entries. See [Parameters, 783](#) for detailed description of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given in [Parameters \(Ch. 6\)](#) of this guide.
2. If LMODES (or LMODESFL)=0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

**FLSPOUT**

## Control for Fluid-Structure Mode Participation Output

Control for fluid-structure mode participation output.

Format:

$$\begin{aligned}
 & \text{FLSPOUT} \quad \left[ \text{FLUIDMP} = \begin{cases} \text{ALL} \\ n_{modes} \\ \text{NONE} \end{cases} \right], \left[ \text{GRIDFMP} = \begin{cases} \text{ALL} \\ setf_{participations} \end{cases} \right] \\
 & \quad \left[ \text{OUTFMP} = \begin{cases} \text{ALL} \\ p_{highest} \\ \text{NOPRINT} \end{cases} \right], \left[ \text{FEPS} = \begin{cases} 1. - 11 \\ epsf \end{cases} \right], \\
 & \quad \left[ \text{ARF} = \begin{cases} 0.95 \\ arf\_v \end{cases} \right], \\
 & \quad \left[ \text{STRUCTMP} = \begin{cases} \text{ALL} \\ m_{modes} \\ \text{NONE} \end{cases} \right], \left[ \text{OUTSMP} = \begin{cases} \text{ALL} \\ q_{highest} \\ \text{NOPRINT} \end{cases} \right] \\
 & \quad \left[ \text{PANELMP} = \begin{cases} \text{ALL} \\ setp_{participations} \\ \text{NONE} \end{cases} \right], \left[ \text{GRIDMP} = \begin{cases} \text{ALL} \\ setg_{participations} \\ \text{NONE} \end{cases} \right] \\
 & \quad \left[ \text{SEPS} = \begin{cases} 1. - 11 \\ epss \end{cases} \right], \left[ \text{ARS} = \begin{cases} 0.95 \\ ars\_v \end{cases} \right] \\
 & \quad \left[ \text{PSORT} = \begin{cases} \text{ABSOLUTE} \\ \text{REAL} \\ \text{IMAGINARY} \end{cases}, \begin{cases} \text{DESCENDING} \\ \text{ASCENDING} \end{cases} \right], \left[ \text{O2E} = \begin{cases} \text{YES} \\ \text{NO} \end{cases} \right]
 \end{aligned}$$

Examples:

|         |  |
|---------|--|
| SET 23  | = ROOF, DRIVERSD   |
| SET 211 | = 1023, 4069, 56790  |
| FLSPOUT | FLUIDMP = 30                    STRUCTMP = 40,                    OUTSMP = 30, |
|         | PANELMP = 23                    GRIDMP = 211                                   |

| Descriptor | Meaning  |
|------------|--|
| FLUIDMP    | Requests fluid participation calculation of fluid response on selected fluid points.   |
| ALL        | Requests that all the fluid modes extracted be used.   |
| n          | Requests that up to the first n fluid modes be used.   |
| NONE       | Requests no participation calculation.   |
| GRIDFMP    | Requests inclusion or exclusion of specific fluid grids to be used in all the requested types of participation calculations. These are also the fluid grids that can be referred to on plot and .op2 tables. |
| ALL        | Requests inclusion in all the requested types of the participation calculations of all fluid points.   |
| setf       | Case Control set ID listing a selected set of fluid grids to be used in all the requested types of participation calculations.   |
| OUTFMP     | Requests the FLUID FLUIDMP participation factors to be output for print.   |
| ALL        | Requests that all FLUID FLUIDMP participation factors to be output for print.  |
| p          | Requests the p highest FLUIDMP participation factors to be output.   |
| NOPRINT    | Produces tables for plotting but do not print any results.   |
| FEPS       | Filters threshold for fluid participation.   |
| epsf       | Threshold value.   |
| ARF        | Acceptance ratio for fluid participation.  |
| arf_v      | Fluid participation values $< arf\_v * \text{max\_value}$ in a column of the output matrix will be set to zero.  |
| STRUCTMP   | Requests structural, load, and panel participation calculations on the selected fluid points. FLUIDMP must be specified for this command to become active.   |
| ALL        | Requests that all the structural modes extracted be used.  |
| m          | Requests that up to the first m structural modes be used.  |
| NONE       | Requests no participation calculation.   |
| OUTSMP     | Requests that structural STRUCTMP participation factors to be output for print.  |
| ALL        | Request that all STRUCTMP participation factors be output.   |
| q          | Requests that the q highest STRUCTMP participation factors be output.  |
| NOPRINT    | Produces tables for plotting but does not print any results.   |
| PANELMP    | Requests inclusion or exclusion of panel participation calculations on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active.                             |
| ALL        | Requests all panels defined be included in the participation calculations on the selected fluid points.  |

| Descriptor | Meaning   |
|------------|---|
| setp       | Case Control set ID listing selected panels for panel participation calculations on the selected fluid points.<br>the set consists of the character names of the panels (new V2001)               |
| NONE       | Requests exclusion from the participation calculations.   |
| GRIDMP     | Requests inclusion or exclusion of a structural panel grid participation calculation on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active. |
| ALL        | Requests, for panels selected, that each and every individual panel grid be included as a separate calculation in the participation calculations on selected fluid points.                        |
| setg       | Case Control set ID listing structural panel grids for grid mode participation on the selected fluid points.  |
| NONE       | Requests exclusion from the participation calculations.   |
| SEPS       | Filter threshold for structure participation.   |
| epss       | Threshold value.  |
| ARS        | Acceptance ratio for structure related fluid participation.   |
| ars_v      | Structure fluid participation values $< arf_v * \text{max\_value}$ in a column of the output matrix will be set to zero.  |
| PSORT      | Requests type of sort.  |
| O2E        | Controls generation of tables of mode participation versus natural frequency for excitation frequencies. These tables are accessible in XYPILOT.  |

### Remarks:

1. This entry represents a collection of PARAM,name,value entries and must appear above the subcase level. See [Parameters, 783](#) for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given in [Parameters](#) (Ch. 6) on this guide.
2. If  $n$ ,  $m$ ,  $p$ , or  $q$  are greater than the number computed, Nastran will invoke the ALL option for the current value.
3. PSORT values must occur in pairs such as (ABSOLUTE,DESCENDING).
4. The underlined item in the {} braces give the value of the keyword if the keyword and its descriptors are omitted from this entry. For example, if FLUIDMP is omitted from the FLSPOUT entry, then no fluid mode participation will be computed (unless a PARAM,FLUIDMP,value explicitly appears in a subcase or [Bulk Data Entries, 1117](#)).

**FLSTCNT**

## Miscellaneous Fluid-Structure Control Parameters

Control for fluid-structure symmetry and force requests.

Format:

$$\begin{aligned}
 FLSTCNT & \left[ ACSYM = \begin{cases} YES \\ NO \end{cases} \right], \left[ ACOUT = \left\{ \begin{array}{c} PEAK \\ RMS \\ PEAKINT \\ RMSINT \end{array} \right\} \right] \\
 & \left[ PREFDB = \begin{cases} 1.0 \\ prp \end{cases} \right], \left[ ASCOUP = \begin{cases} YES \\ NO \end{cases} \right] \\
 & \left[ SKINOUT = \begin{cases} NONE \\ PUNCH \\ PRINT \\ ALL \end{cases} \right]
 \end{aligned}$$

Example(s):

FLSTCNT      ACSYM = YES      ACOUT = RMS

| Descriptor    | Meaning  |
|---------------|--|
| ACSYM         | Requests symmetric or nonsymmetric solution for fluid-structure analysis.  |
| YES           | Requests symmetrized coupled fluid-structure analysis.   |
| NO            | Requests no symmetric coupled fluid-structure analysis.  |
| ACOUT         | Requests peak or rms for output to be used with the FORCE request.   |
| PEAK, PEAKINT | Requests peak value output to be used with the FORCE request.  |
| RMS, RMSINT   | Requests rms value output to be used with the FORCE request.   |
| PREFDB        | Specifies the peak reference pressure.   |
| prp           | Value for the peak reference pressure.   |
| ASCOUP        | Request a coupled or uncoupled fluid-structure analysis.   |
| YES           | Request a coupled fluid-structure analysis.  |
| NO            | Request a uncoupled fluid-structure analysis.  |
| SKINOUT       | Request that sets of grid point and element lists be output for both the fluid and structure at the fluid-structure interface. |
| NONE          | Requests no output of sets.  |
| PUNCH         | Requests set output to punch file (.pch) only.   |

| Descriptor | Meaning  |
|------------|--|
| PRINT      | Requests set output to .f06 file only.           |
| ALL        | Requests set output to both .pch and .f06 files. |

**Remarks:**

1. This entry represents a collection of PARAM,name,value entries. See [Parameters, 783](#) for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given under the parameter description in this guide.
2. Options with suffix INT generates INTENSITY, instead of ACCELERATION. Unlike ACCELERATION which is complex, INTENSITY is a scalar quantity.

**FLUX****Heat Transfer Gradient and Flux Output Request**

Requests the form and type of gradient and flux output in heat transfer analysis.

**Format:**

$$FLUX \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

FLUX=ALL

FLUX (PUNCH, PRINT)=17

FLUX=25

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |      |   |
|------|---|
| ALL  | Flux for all elements will be output.   |
| NONE | Flux for no elements will be output.  |
| n    | Set identification of a previously appearing SET command. Only fluxes of elements with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. FLUX=ALL in SOL 159 may produce excessive output.
2. FLUX=NONE overrides an overall request.

**FMETHOD**

## Flutter Analysis Method Parameter Selection

Selects the parameters to be used by the aerodynamic flutter analysis.

**Format:**

FMETHOD=n

**Example:**

FMETHOD=72

| Describer | Meaning   |
|-----------|---|
| n         | Set identification number of a FLUTTER Bulk Data entry (Integer > 0). |

**Remarks:**

1. An FMETHOD command is required for flutter analysis.
2. A CMETHOD command is also required for the K-method of flutter analysis.
3. If this entry is being used in SOL 200 in conjunction with flutter design conditions, the METHOD selected on the FLUTTER Bulk Data entry must be "PK" or "PKNL".

**FORCE****Element Force Output or Particle Velocity Request**

Requests the form and type of element force output, or particle velocity output, in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.

Format:

$$FORCE \left[ \begin{array}{c} \left[ \begin{array}{c} SORT1 \\ SORT2 \end{array} \right], \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right], \left[ \begin{array}{c} REAL \text{ or } IMAG \\ PHASE \end{array} \right], \left[ \begin{array}{c} CENTER \\ CORNER \text{ or } BILIN \\ SGAGE \\ CUBIC \end{array} \right], \\ \left[ \begin{array}{c} PSDF, ATOC, CRMS \\ \text{or } RALL \end{array} \right], \left[ \begin{array}{c} RPRINT \\ NORPRINT, RPUNCH \end{array} \right] \end{array} \right] = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

```
FORCE=ALL
FORCE (REAL, PUNCH, PRINT)=17
FORCE=25
FORCE (SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
FORCE (PRINT, RALL, NORPRINT)=ALL
```

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each element type.  |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

| Descriptor         | Meaning  |
|--------------------|--|
| REAL or IMAG       | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.   |
| PHASE              | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.   |
| PSDF               | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8. |
| ATOC               | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.        |
| CRMS               | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.                       |
| RALL               | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.                                    |
| CENTER             | Output CQUAD4, CQUADR, and CTRIAR element forces at the center only. The default for CQUAD4 is CENTER. The default for CQUADR and CTRIAR is CORNER.  |
| CORNER or<br>BILIN | Output CQUAD4, QUADR, and CTRIAR element forces at the center and at the grid points using strain gage approach with bilinear extrapolation.   |
| SGAGE              | Output CQUAD4 element forces at the center and at the grid points using strain gage approach.  |
| CUBIC              | Output CQUAD4 element forces at the center and at the grid points using cubic bending correction.  |
| RPRINT             | Writes random analysis results in the print file (Default).  |
| NORPRINT           | Disables the writing of random analysis results in the print file.   |
| RPUNCH             | Writes random analysis results in the punch file.  |
| ALL                | Forces for all elements will be output.  |
| n                  | Set identification of a previously appearing SET command. Only forces of elements with identification numbers that appear on this SET command will be output (Integer > 0).  |
| NONE               | Forces for no elements will be output.   |

**Remarks:**

1. ALL should not be used in a transient problem.
2. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.

3. ELFORCE is an alternate form and is entirely equivalent to FORCE.
4. FORCE=NONE overrides an overall request.
5. If PARAM,SPARSEDRL,NO is specified, then to request force output on damping elements in modal frequency response analysis (e.g., SOL 111), the mode displacement method (PARAM,DDRMM,-1) must be selected.

Force output on damping elements is not available in transient response analysis.
6. In nonlinear transient analysis, the FORCE request is ignored for nonlinear elements in SOL129. In SOL400 with "ANALYSIS=NLTTRAN" Element FORCE output for the CWELD/CFAST elements is available. All other elements capable of force output such as the CBEAM, CQUAD4, etc., will not produce nonlinear transient force output.

If "PARAM, OLDWELD, YES" is specified, then the CWELD/CFAST elements will not produce with "ANALYSIS=NLTTRAN" Element FORCE output.
7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE Case Control commands. Consequently, options specified in subcases other than the first subcase will be ignored. The BILIN or CORNER option is not available for advanced nonlinear elements and nonlinear material or composite elements. The default option will be applied to those elements.
  - a. If the STRESS command is specified in the first subcase, then the option on the STRESS command is used in all subcases with STRESS, STRAIN, and FORCE commands.
  - b. If the STRAIN command and no STRESS command is specified in the first subcase, then the option on the STRAIN command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
  - c. If the FORCE command and no STRESS or STRAIN commands is specified in the first subcase, then the option on the FORCE command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
  - d. If STRESS, STRAIN, and FORCE commands are not specified in the first subcase, then the CENTER option is used in all subcases containing STRESS, STRAIN, and FORCE commands.
8. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
9. In general, for nonlinear elements, force output is not available. For CGAP, CELASi, or CBUSH elements in nonlinear analysis, use the STRESS in NLSTRESS command to obtain force output.
10. Forces and moment output for line elements are total forces and for shell elements they are forces per unit length.
11. The FORCE request is ignored for solid elements in all solution sequences as it has no meaning for solids.

**FREQUENCY****Frequency Set Selection**

Selects the set of forcing frequencies to be solved in frequency response problems.

**Format:**

**FREQUENCY=n**

**Example:**

**FREQUENCY=17**

| Describer | Meaning   |
|-----------|---|
| n         | Set identification number of FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5 Bulk Data entries. (Integer > 0) |

**Remarks:**

1. A frequency set selection is required for a frequency response problem.
2. A frequency set selection is required for transient response by Fourier methods (SOL 146).
3. All FREQ*i* entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQ*i* entries, respectively.

4. If there are multiple frequency response subcases, improved performance can be obtained by making the all the subcases that share the same FREQUENCY ID contiguous. For SOL 200, it is an error if there are noncontiguous subcases that have the same boundary condition and FREQUENCY ID.
5. When a FATIGUE case control is also present to perform random vibration fatigue analysis using SOLs 108 and 111, only one FREQUENCY case control set is allowed for all transfer function SUBCASEs and should be above the SUBCASE level or only present in 1st SUBCASE. The FREQUENCY set cannot change from SUBCASE to SUBCASE or a fatal error is issued.

**FRF**

Frequency Response Function (FRF) Generation and/or FRF Based Assembly (FBA) Specification

Specifies the information needed for FRF generation and/or the FBA process. SOLs 108 and 111 only.

Format:

$$\begin{aligned}
 & \text{FRF} \left[ \left( \begin{array}{c} \text{GEN} \\ \text{ASM} \\ \text{GENASM} \end{array} \right) \right] [\text{COMPID} = \text{cmpid}] [\text{COMPNAME} = \text{cmpname}] \\
 & [\text{CONNPTS} = \text{setid}] \left[ \text{XITOUT} = \left\{ \begin{array}{c} \text{UNIT} \\ \text{UNITALL} \\ \text{USER} \\ \text{USERTOTL} \end{array} \right\} \right] \left[ \text{ASMOUP} = \left\{ \begin{array}{c} \text{COMP} \\ \text{ALL} \\ \text{ASSEMBLY} \\ \text{CONNINFO} \\ n \\ cname \end{array} \right\} \right] \left[ \text{LOADLBL} = \left\{ \begin{array}{c} \text{STD} \\ \text{ALT} \\ \text{ALTX} \end{array} \right\} \right] \\
 & \left[ \left( \begin{array}{c} \text{DB} \\ \text{OP2} = \text{unit} \end{array} \right) \right]
 \end{aligned}$$

$$\left\{ \begin{array}{l} \text{ICFGEN= } \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{compname} \end{array} \right\} \\ \text{ICFUSE= } \left\{ \begin{array}{c} n \\ \text{compname} \end{array} \right\} \\ \text{ICFAUTO= } \left\{ \begin{array}{c} n \\ \text{compname} \end{array} \right\} \end{array} \right\} = \left\{ \begin{array}{c} \text{ICFDDB} \\ \text{ICFOP2}=\{\text{icfunit}\} \end{array} \right\}$$

Examples:

```

FRF
FRF (COMPID = 10 COMPNAME = WING CONNPTS = 1000 LOADLBL = ALT)
FRF (COMPID = 20 COMPNAME = STRUT CONNPTS = 2000 OP2 = 25)
FRF (COMPID = 30 COMPNAME = BODY CONNPTS = 300 XITOUT = UNITALL)
FRF (ASM)
FRF (ASM LOADLBL = ALTX)
FRF (ASM ASMOUP = ALL)
FRF (GENASM COMPID = 50 COMPNAME = SUSPNSN CONNPTS = 200)

```

```

FRF (ASM ICFGEN = ALL)
FRF (ASM ICFUSE = 100)
FRF (ASM ICFGEN = 200 ICFOP2 = 33)
FRF (ASM ICFUSE = -1 ICFOP2 = 33)
FRF (ASM ICFAUTO = 100)
FRF (ASM ICFAUTO = -5 ICFOP2 = 31)

```

| Descriptor                | Meaning   |
|---------------------------|---|
| GEN<br>(Default)          | Generate the FRFs for the specified component. See Remarks 3. and 4., and the Examples in Remark 13.  |
| ASM                       | Compute the FRFs of an assembly of components from the FRFs of the individual components. See Remarks 5., 9., 10. and 11., and Examples 2 and 3 in Remark 13.   |
| GENASM                    | Generate the FRFs for the specified component and follow it by computing the FRFs of an assembly of components from the FRFs of the individual components. See Remarks 4., 9., 10. and 11., and Examples 4 and 5 in Remark 13.  |
| COMPID = <i>cmpid</i>     | <i>cmpid</i> (Integer > 0) is the identification number of the component whose FRFs are to be generated. See Remarks 3. through 7., and Examples 2 through 5 in Remark 13.  |
| COMPNAME = <i>cmpname</i> | <i>cmpname</i> (up to eight characters) is the name of the component whose FRFs are to be generated. See Remarks 3. through 7., and Examples 2 through 5 in Remark 13.  |
| CONNPTS = <i>setid</i>    | <i>setid</i> (integer > 0) refers to the set that defines the points at which the FRF component specified by the COMPID/COMPNAME keywords is to be connected during a subsequent FRF based assembly (FBA) process. Only those points that are defined in this set (and no others) will be considered for connection during the FBA process. See Remarks 7., 12.(c) and 12.(d), and Examples 2 through 5 in Remark 13. |
| XITOUT = UNIT             | Output the FRF results only for those unit excitations that are specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control request. See Remark 8.   |
|                           | The output for each of the above excitations is identified by a separate subcase. The IDs of these subcases are numbered consecutively starting from 1.   |

| Descriptor       | Meaning   |
|------------------|---|
| XITOUT = UNITALL | <p>Output FRF results not only for unit excitations specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control command, but also for unit excitations that are internally applied automatically by the program at the connection points of the FRF component(s). See Remarks 8. and 12.(c), and Example 2 in Remark 13.</p> <p>The output for each of the above excitations is identified by a separate subcase. The IDs of these subcases are numbered consecutively starting from 1.</p>  |
| XITOUT = USER    | <p>Output the FRF results for the following excitations implied by the DLOAD Case Control request:</p> <ul style="list-style-type: none"> <li>a. A separate excitation for each individual DOF that has a non-zero load value specified for it</li> <li>b. An excitation representing the total load</li> </ul> <p>Thus, if a DLOAD Case Control request involves non-zero load values on N DOFs, then this request gives results for (N+1) excitations, with the first N such excitations representing individual and separate loads on the N DOFs and the (N+1)th excitation representing the total load. See Remark 8.</p> <p>The output for each of the above (N+1) excitations is identified by a separate coded subcase ID of the form xxxx yyyy. Here xxxx is the user subcase ID corresponding to the DLOAD under consideration. For the first N excitations, yyyy has values ranging from 1 through N (with leading zeros where appropriate). For the (N+1)th excitation representing the total load, the coded subcase ID is of the form xxxx9999.</p> <p>Because of the above coded numbering scheme, when XITOUT = USER is specified (or assumed; see Remark 8), the program will not allow any user subcase ID to exceed 9999. If it does, the program terminates the job with an appropriate fatal message.</p> |

| Descriptor                 | Meaning   |
|----------------------------|---|
| XITOUT = USERTOTL          | <p>Output the FRF results for the single excitation representing the total load implied the DLOAD Case Control request. This corresponds to the (N+1)th excitation mentioned earlier. See Remark 8.</p> <p>The output for this single excitation representing the total load is identified by the coded subcase ID of the form xxxx9999 where xxxx is the user subcase ID corresponding to the DLOAD under consideration.</p> <p>Because of the above coded numbering scheme, when XITOUT = USERTOTL is specified, the program will not allow any user subcase ID to exceed 9999. If it does, the program terminates the job with an appropriate fatal message.</p>   |
| ASMOUT = CONNINFO          | In the FBA process, terminate the job after generating the FRF component connection information output without performing any further FRF assembly calculations.  |
| ASMOUT = COMP<br>(Default) | In the FBA process, output the FRF results for all of the individual FRF components comprising the assembly. See Remarks 10. and 11.  |
| ASMOUT = ALL               | In the FBA process, output the FRF results not only for all of the individual FRF components comprising the assembly, but also for the assembled FRF configuration considered as a separate entity. See Remarks 10. and 11., and Example 3 in Remark 13. See also Remark 2.   |
| ASMOUT = ASSEMBLY          | In the FBA process, output the FRF results only for the assembled FRF configuration considered as a separate entity. This is equivalent to specifying ASMOUT = 0 (described later). See Remarks 2. and 10.  |
| ASMOUT = <i>n</i>          | <p><i>n</i> is an integer with the following meanings:</p> <p><i>n</i> = 0</p> <p>In the FBA process, output the FRF results only for the assembled FRF configuration considered as a separate entity. This is equivalent to specifying ASMOUT = ASSEMBLY (described earlier). See Remarks 2. and 10.</p> <p><i>n</i> &gt; 0</p> <p>In the FBA process, output the FRF results only for those FRF components of the assembly whose IDs are specified by SET ID <i>n</i>.</p> <p><i>n</i> &lt; 0</p> <p>In the FBA process, output the FRF results only for that single FRF component of the assembly whose ID is given by <math> n </math>.</p> <p>In the FBA process, output the FRF results only for that single FRF component of the assembly whose name is given by <i>cname</i>.</p> |
| ASMOUT = <i>cname</i>      |   |

| Descriptor                 | Meaning   |
|----------------------------|---|
| LOADLBL = STD<br>(Default) | The load labels in the output in the FRF and FBA jobs explicitly identify the grid (or scalar) point and its component where the load is applied.   |
| LOADLBL = ALT              | <p>The load labels in the output in the FRF and FBA jobs identify the grid (or scalar) point and its component where the load is applied by using the following notation:</p> <p>GGGGGGGG:+C</p> <p>where GGGGGGGG is the grid (or scalar) point ID and C indicates the component where the load is applied. C may have the following values:</p> <ul style="list-style-type: none"> <li>X Indicates grid point component 1 or a scalar point</li> <li>Y Indicates grid point component 2</li> <li>Z Indicates grid point component 3</li> <li>RX Indicates grid point component 4</li> <li>RY Indicates grid point component 5</li> <li>RZ Indicates grid point component 6</li> </ul> |
| LOADLBL = ALTX             | Same as LOADLBL = ALT except that the load labels also identify whether the load applied is a unit load or a user load.   |
| DB<br>(Default)            | Store the FRF matrices and other information on the database. See Examples 2 and 4 in Remark 13.  |
| OP2 = <i>unit</i>          | Store the FRF matrices and other information on an OUTPUT2 file whose Fortran unit number is given by <i>unit</i> (integer > 0). See Examples 3 and 5 in Remark 13.   |
| ICFGEN = ALL               | Generate ICF information in the FBA process for all of the FRF components of the assembly   |
| ICFGEN = n                 | <p>n is a non-zero integer with the following meanings: n &gt; 0</p> <p>Generate ICF information in the FBA process only for those FRF components of the assembly whose IDs are specified by SET ID n,</p> <p>n &lt; 0</p> <p>Generate ICF information in the FBA process only for that single FRF component of the assembly whose ID is given by  n .</p>  |
| ICFGEN = compname          | FRF component whose name is given by compname.  |

| Descriptor              | Meaning   |
|-------------------------|---|
| ICFUSE = n              | n is a non-zero integer with the following meanings:<br><br>n > 0<br><br>Use ICF information by employing in the FBA process a Configuration that consists of only those FRF components whose IDs are specified by SET ID n.<br><br>n < 0<br><br>Use ICF information by employing in the FBA process a Configuration that consists of only that single FRF component whose ID is given by  n . Configuration that consists of only that single FRF component whose name is given by compname. |
| ICFAUTO = n             | n is a non-zero integer with the following meanings:<br><br>n > 0<br><br>First generate and then use ICF information in the FBA process only for those FRF components whose IDs are specified by SET ID n.<br><br>n < 0<br><br>First generate and then use ICF information in the FBA process only for that single FRF component whose ID is given by  n . only for that single FRF component whose name is given by compname.  |
| ICFDB (Default)         | ICF information is to be stored or is resident on the database.   |
| ICFOP2 = <i>icfunit</i> | ICF information is to be stored or is resident on an OUTPUT2 file whose Fortran unit number is given by icfunit (integer > 0).  |

**Remarks:**

1. This command is supported only in SOLs 108 and 111.
2. A component ID of 0 is assigned to the assembled FRF configuration resulting from the FBA process.
3. The COMPNAME keyword must be specified if the COMPID keyword is specified and vice versa.

4. If the COMPID/COMPNAME keywords are specified along with the GEN/GENASM keyword, then it implies that the FRFs computed for the specified component are employed in a subsequent FBA process. In this case, the FRF generation results will be saved on the specified medium, and the .asm (assembly punch) file will be generated and saved with a single FRFCOMP Bulk Data entry in it for subsequent use in an FBA process.
5. If the COMPID/COMPNAME keywords are not specified with the GEN/GENASM keyword, then it implies that the FRFs computed are for a single shot configuration with no subsequent FBA process involved. In this case, the GENASM keyword is equivalent to the GEN keyword.
6. The COMPID/COMPNAME keywords are ignored if the ASM keyword is specified.
7. The CONNPTS keyword must be specified if the COMPID/COMPNAME keywords are specified. It is ignored otherwise.
8. If the XITOUT keyword is not specified, the default of XITOUT = UNIT is assumed if there is no DLOAD Case Control request and the default of XITOUT = USER is assumed if there is a DLOAD Case Control request. If the user specifies XITOUT = USER or XITOUT = USERTOTL, but there is no DLOAD Case Control request, the program issues a warning message and assumes XITOUT = UNIT.
9. If the ASM/GENASM keyword is specified, the resulting FBA process will generate a connection information table in the .f06 file, indicating the relationship between the internal point IDs of the assembled FRF configuration (referred to as component 0 as indicated in Remark 2.) and the external point IDs of the associated FRF components.
10. The normal output from an FBA process run, implied by the default of ASMOUP = COMP, gives the results for the individual FRF components that comprise the assembled FRF configuration. If output is also desired for the assembled configuration as a separate entity (component 0 as indicated in Remark 2.), then ASMOUP = ALL must be specified in the FRF command to obtain the expanded output. However, in this case, the output for component 0 will be limited to displacements, velocities, and accelerations, and these will be output in terms of the internal point IDs mentioned in Remark 9. See Example 3 in Remark 13.
11. The ASMOUP keyword is ignored if the GEN keyword is specified.
12. The generation of FRFs for a component and their use in a subsequent FBA process using the FRF Case Control command involves running a standard SOL 108 or SOL 111 job, with the following additional data:
  - a. The DOFs where loads are to be applied must be specified either indirectly via the DLOAD Case Control command and/or directly via the FRFXIT/FRFXIT1 Bulk Data entries. The DLOAD Case Control command points to appropriate Bulk Data loading entries. All DOFs with nonzero load values will have unit loads applied to them. The FRFXIT entry permits specification of unit load for a single DOF with a label. The FRFXIT1 entry permits specification of unit loads at multiple DOFs.
  - b. There is no requirement that unit loading data be defined for every component for which FRFs are generated, since some components in a configuration may not have any loads applied to them.

- c. Regardless of whether an FRF component has unit loads explicitly specified for it, as in Remark 12(a) or not, as in Remark 12(b), the program will internally apply unit loads automatically at all DOFs for all connection points comprising the set referenced by the CONNPTS keyword. This ensures that correct results are obtained from subsequent FBA processes.
  - d. The specific points at which FRFs are computed in an FRF generation run consist of the following:
    - All points specified via DISP, VELO, and ACCE requests
    - All points associated with elements for which STRESS/FORCE requests are specified
    - All points at which unit loads are applied (as per the scheme indicated in Remark 12(a))
    - All points comprising the set referenced by the CONNPTS keyword
    - All grid points referenced in PLOTEL Bulk Data entries
  - e. It is assumed in an FBA process that the FRFs of all of the FRF components have been generated at the same forcing frequencies, and that these are also the forcing frequencies at which the FBA process is to be performed. As a result, the FBA process derives these forcing frequencies from the saved data of the first of the FRF components being assembled, and uses them in the FBA process.
- In order to ensure the validity of the FBA process, the program checks to make sure that all of the FRF components have been generated using the same number of forcing frequencies and further that all of these forcing frequencies are the same for all of the FRF components. If both of these conditions are not met, the program terminates the job with an appropriate fatal message.
13. The following examples illustrate details of job setups for FRF generation and the subsequent FBA process for various scenarios.

Example 1 involves FRF generation for a single shot configuration without any FBA process.

Examples 2 through 5 assume that there are three components – 10, 20 and 30 – for which FRFs are to be generated (frgen10.dat, frfgen20.dat and frgen30.dat) and that the FRFs of these components are to be subsequently assembled in an FBA process to obtain the FRFs of the assembled configuration.

Example 6 illustrates an FBA process involving the assembly of two components – 10 and 20 – whose FRFs are generated by Nastran, with a third FRF component – 40 – whose FRFs have been generated from test, with its FRF and other information resident on an Universal File (UF).

Examples 2, 3 and 6 illustrate the use of the ASM option for the FBA process while Examples 4 and 5 illustrate the use of the GENASM option for the FBA process.

Examples 7 through 13 deal with an airplane model comprising five FRF components, namely, fuselage (1), horizontal tail (2), vertical tail (3), inboard wings (4) and outboard wings (5). Example 7 illustrates the generation of the FRFs for these five components while the other examples are FBA jobs that illustrate the generation and usage of ICFs (inter component forces) for several scenarios using the ICFGEN, ICFUSE and ICFAUTO keywords of the FRF Case Control command, employing both database and OUTPUT2 usage.

Loading data must be defined for Example 1 either via FRFXIT/FRFXIT1 Bulk Data entries or via the DLOAD Case Control request. For all other examples, loading data may be defined as desired either in the FRF generation runs or in the FBA process or both.

**Example 1. Generate FRFs for a Single Shot Configuration with No Subsequent FBA Process**

- FRF Generation Job
  - Case Control Requirement
  - FRF

Loading data must be defined either via the DLOAD Case Control command and/or via FRFXIT/FRFXIT1 Bulk Data entries.

**Example 2. Generate FRFs for Components 10, 20, and 30 Using the DB Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option**

- FRF Generation Jobs

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200)

FRF component 30: FRF (COMPID = 30 COMPNAME = COMP30 CONNPTS = 300)

For these jobs, scr = no should be specified on the Nastran job command lines to ensure that the databases are saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

- FBA Job

- File Management Section (FMS) Requirement

ASSIGN dbname10 = 'frfgen10.MASTER'

ASSIGN dbname20 = 'frfgen20.MASTER'

ASSIGN dbname30 = 'frfgen30.MASTER'

DBLOCATE DATABLK = (FRFDB) LOGICAL = dbname10

DBLOCATE DATABLK = (FRFDB) LOGICAL = dbname20

DBLOCATE DATABLK = (FRFDB) LOGICAL = dbname30

(Note: All of the data blocks stored on the databases from the FRF generation runs have the same common name of FRFDB.)

- Case Control Requirement

FRF (ASM XITOUT = UNITALL)

The XITOUT = UNITALL request gives output for all unit excitations (both user specified unit loads and internally applied unit loads).

- Bulk Data Requirement

**Example 2. Generate FRFs for Components 10, 20, and 30 Using the DB Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option**

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

INCLUDE 'frfgen30.asm'

**Example 3. Generate FRFs for Components 10, 20, and 30 Using the OP2 Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2 = 'frfgen10\_op2' UNIT = 25 DELETE

ASSIGN OUTPUT2 = 'frfgen20\_op2' UNIT = 26 DELETE

ASSIGN OUTPUT2 = 'frfgen30\_op2' UNIT = 27 DELETE

For these jobs, scr = yes may be specified on the Nastran job command lines since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

FRF component 30: FRF (COMPID = 30 COMPNAME = COMP30 CONNPTS = 300 OP2 = 27)

■ FBA Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2 = 'frfgen10\_op2' UNIT = 25

ASSIGN INPUTT2 = 'frfgen20\_op2' UNIT = 26

ASSIGN INPUTT2 = 'frfgen30\_op2' UNIT = 27

- Case Control Requirement

FRF (ASM ASMOUT = ALL)

The ASMOUT = ALL request gives output from the FBA process not only for FRF components 10, 20 and 30, but also for the assembled configuration as a separate entity (component 0) as indicated in Remark 10.

- Bulk Data Requirement

**Example 3. Generate FRFs for Components 10, 20, and 30 Using the OP2 Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option**

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

INCLUDE 'frfgen30.asm'

**Example 4. Generate FRFs for Components 10 and 20 Using the DB Option and Subsequently Assemble their FRFs with Those of Component 30 Using the DB and GENASM Options to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200)

For these jobs, scr = no should be specified on the Nastran job command line to ensure that the databases are saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ Combined FRF Generation and FBA Job

- File Management Section (FMS) Requirement

ASSIGN dbname10 = 'frfgen10.MASTER'

ASSIGN dbname20 = 'frfgen20.MASTER'

DBLOCATE DATABLK=(FRFDB) LOGICAL=dbname10

DBLOCATE DATABLK=(FRFDB) LOGICAL=dbname20

(Note: All data blocks stored on the databases from the FRF generation runs have the same common name of FRFDB.)

For this job, scr = no should be specified on the Nastran job command line to ensure that the database for FRF component 30 is saved for subsequent use by the FBA process.

- Case Control Requirement

FRF (GENASM COMPID = 30 COMPNAME = COMP30 CONNPTS = 300)

- Bulk Data Requirement

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

**Example 5. Generate FRFs for Components 10 and 20 Using the OP2 Option and Subsequently Assemble their FRFs with Those of Component 30 Using the OP2 and GENASM Options to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT2 = 'frfgen10_op2' UNIT = 25 DELETE
```

```
ASSIGN OUTPUT2 = 'frfgen20_op2' UNIT = 26 DELETE
```

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

For these jobs, scr = yes may be specified on the Nastran job command line since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ Combined FRF Generation and FBA Job

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'frfgen10_op2' UNIT = 25
```

```
ASSIGN INPUTT2 = 'frfgen20_op2' UNIT = 26
```

```
ASSIGN OUTPUT2 = 'frfgen30_op2' UNIT = 27 DELETE
```

- Case Control Requirement

FRF (GENASM COMPID = 30 COMPNAME = COMP30 CONNPTS = 300 OP2 = 27)

- Bulk Data Requirement

The following INCLUDEs are required.

```
INCLUDE 'frfgen10.asm'
```

```
INCLUDE 'frfgen20.asm'
```

**Example 6. Generate FRFs for Components 10 and 20 Using the OP2 Option and Subsequently Assemble their FRFs with Those of Test FRF Component 40 Using the ASM Option to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2 = 'ffrfg10\_op2' UNIT = 25 DELETE

ASSIGN OUTPUT2 = 'ffrfg20\_op2' UNIT = 26 DELETE

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

For these jobs, scr = yes may be specified on the Nastran job command line since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ FBA Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2 = 'ffrfg10\_op2' UNIT = 25

ASSIGN INPUTT2 = 'ffrfg20\_op2' UNIT = 26

ASSIGN UNVFILE = 'ffrfg40\_unv' UNIT = 28 \$ Universal File

- Case Control Requirement

FRF (ASM)

- Bulk Data Requirement

FRFCOMP,40,TSTCMP40,UF,28

The following INCLUDEs are also required.

INCLUDE 'ffrfg10.asm'

INCLUDE 'ffrfg20.asm'

### Example 7. Generate FRFs for Components 1 through 5 Using the OP2 Option

- File Management Section (FMS) Requirement

```
FRF Component 1:ASSIGN OUTPUT2 = 'fuselage_op2' UNIT=25 DELETE
FRF Component 2:ASSIGN OUTPUT2 = 'hor_tail_op2' UNIT=26 DELETE
FRF Component 3:ASSIGN OUTPUT2 = 'ver_tail_op2' UNIT=27 DELETE
FRF Component 4:ASSIGN OUTPUT2 = 'ib_wings_op2' UNIT=28 DELETE
FRF Component 5:ASSIGN OUTPUT2 = 'ob_wings_op2' UNIT=29 DELETE
```

- Case Control Requirement

```
FRF Component 1:FRF (COMPID = 1 COMPNAME = FUSELAGE CONNPTS =
1000 OP2=25)
FRF Component 2:FRF (COMPID = 2 COMPNAME = HOR_TAIL CONNPTS =
1000 OP2=26)
FRF Component 3:FRF (COMPID = 3 COMPNAME = VER_TAIL CONNPTS =
1000 OP2=27)
FRF Component 4:FRF (COMPID = 4 COMPNAME = IB_WINGS CONNPTS =
1000 OP2=28)
FRF Component 5:FRF (COMPID = 5 COMPNAME = OB_WINGS CONNPTS =
1000 OP2=29)
```

These jobs automatically generate .asm files for subsequent use by the FBA process.

For these jobs, scr = yes may be specified on the Nastran job command lines since there is no need for the databases to be saved at the end of the jobs.

### Example 8. Generate ICFs on the Database for All Five FRF Components of Example 7 (Step 1 of Two-Step Process)

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25
ASSIGN INPUTT2 = 'hor_tail_op2' UNIT=26
ASSIGN INPUTT2 = 'ver_tail_op2' UNIT=27
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29
```

- Case Control Requirement

**Example 8. Generate ICFs on the Database for All Five FRF Components of Example 7 (Step 1 of Two-Step Process)**

FRF (ASM ICFGEND = ALL)

- Bulk Data Requirement

```
INCLUDE 'fuselage.asm'
INCLUDE 'hor_tail.asm'
INCLUDE 'ver_tail.asm'
INCLUDE 'ib_wings.asm'
INCLUDE 'ob_wings.asm'
```

For this job, scr = no should be specified on the Nastran job command line since the database containing the ICF information needs to be saved for use in a subsequent FBA job (Example 9).

**Example 9. Use ICFs of Example 8 for An Assembly Configuration Consisting of FRF Components 1, 4 and 5 (Step 2 of Two-Step Process)**

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29
ASSIGN ICFDATA = 'example8.MASTER'
DBLOCATE (DATABLK = ICFDB) LOGICAL = ICFDATA
```

- Case Control Requirement

```
SET 100 = 1,4,5
FRF (ASM ICFUSE = 100)
```

- Bulk Data Requirement

```
INCLUDE 'fuselage.asm'
INCLUDE 'ib_wings.asm'
INCLUDE 'ob_wings.asm'
```

For this job, scr = yes may be specified on the Nastran job command line since there is no need for the database to be saved at the end of the job.

**Example 10. Generate ICFs on an OUTPUT2 File for FRF Components 1, 2 and 3 of Example 7 (Step 1 of Two-Step Process)****■ File Management Section (FMS) Requirement**

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25
ASSIGN INPUTT2 = 'hor_tail_op2' UNIT=26
ASSIGN INPUTT2 = 'ver_tail_op2' UNIT=27
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29
ASSIGN OUTPUT2 = 'icf123_op2' UNIT=33 DELETE
```

**■ Case Control Requirement**

```
SET 100 = 1,2,3
FRF (ASM ICFFGEN = 100 ICFOOP2 = 33)
```

**■ Bulk Data Requirement**

```
INCLUDE 'fuselage.asm'
INCLUDE 'hor_tail.asm'
INCLUDE 'ver_tail.asm'
INCLUDE 'ib_wings.asm'
INCLUDE 'ob_wings.asm'
```

For this job, scr = yes may be specified on the Nastran job command line since there is no need for the database to be saved at the end of the job.

**GPFORCE****Grid Point Force Output Request**

Requests grid point force balance at selected grid points.

**Format:**

$$GPFORCE \left[ \begin{array}{l} PRINT, PUNCH \\ PLOT \end{array} \right] = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

**Examples:**

GPFORCE=ALL  
GPFORCE=17

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL

Grid point force balance for all grid points will be output.

n

Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the grid point force balance output (Integer >0).

**Remarks:**

1. The printing of the grid point forces will be suppressed if PARAM,NOGPF,-1 appears in the Bulk Data.
2. The Bulk Data entry PARAM,NOELOF,+1 will cause the output of the grid point forces to be aligned with the edges of the two-dimensional elements. The default value of -1 will suppress this output. See Remark 4.
3. The Bulk Data entry PARAM,NOELOP,+1 will cause the output of the sum of the forces parallel to the edges of adjacent elements. The default value of -1 will suppress this output. See Remarks 4. and 10.
4. The output of grid point forces aligned with the edges of elements is available for the following elements:

CBAR  
CROD  
CBEAM  
CSHEAR  
CONROD  
CTRIA3  
CQUAD4  
CTUBE

The positive direction for grid point forces aligned with the edges of elements is from the reference point to the load point as indicated on the printed output. See Remark 10.

5. The grid point force balance is computed from linear and nonlinear elements, and includes the sum of applied loads, contact force, thermal loads, MPC forces, rigid elements, general elements, DMIG entries and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), slideline force contributions, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. The following table summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system:

| Contributions Included                       | Contributions Ignored   |
|--|---|
| Applied Loads                                | DMI Forces  |
| SPC Forces                                   | Boundary Loads from Upstream Superelements  |
| Element Elastic Forces                       | Forces on elements with geometric or material nonlinear properties during normal modes analysis in SOL 106; also called linear perturbation analysis. |
| GENEL Forces                                 | Connector elements CFAST, CSEAM, and CWELD.   |
| DMIG Referenced by K2GG Case Control command |   |
| Thermal Loads                                |   |
| MPC and Rigid Element Forces                 |   |
| Contact Forces                               |   |

6. Only the element elastic forces are included when the grid point forces are aligned with the edges of elements. See Remark 10.
7. In inertia relief analysis, the SPCFORCE and applied load output includes both the effect of inertial loads and applied loads.
8. When pressure loads are applied, the GPFDR module uses the discrete load vector and does not include any distributed effects.
9. GPFORCE is not available in SOLs 108, 111 or 129.

10. Grid point force output is available for nonlinear static analysis for SOLs 106 and 400. SOL106 with thermal loading does not back out the thermal load so result will not sum to zero, this is a limitation of SOL106. Contributions from slideline elements are ignored. PARAM,NOELOF and PARAM,NOELOP are not supported in nonlinear and Remarks [2.](#), [3.](#), [4.](#), and [6.](#) do not apply.
11. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.

**GPKE****Grid Point Kinetic Energy Output Request**

Requests the output of the kinetic energy at selected grid points in normal modes analysis only.

Format:

$$GPKE \left[ \begin{array}{l} PRINT \\ NOPRINT \end{array} \right], (PUNCH, THRESH = e) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

GPKE=ALL

GPKE (PRINT, PUNCH)=19

| Descriptor | Meaning   |
|------------|---|
| PRINT      | The printer will be the output medium.  |
| NOPRINT    | Generates, but does not print, grid point kinetic energy output.  |
| PUNCH      | The punch file will be the output medium.   |
| e          | Minimum energy threshold. Only energies above this value will be printed and/or punched.  |
| ALL        | Grid point kinetic energy for all grid points will be output.   |
| n          | Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in output (Integer > 0). |
| NONE       | Grid point kinetic energy for no points will be output.   |

Remarks:

1. Grid point kinetic energy is only available for normal modes analysis.
2. Both PRINT and PUNCH may be requested.
3. GPKE=NONE overrides an overall output request.
4. For models using the lumped mass formulation, the grid point kinetic energy can be used to examine the distribution of kinetic energy among the grid points. It is computed as:

$$E_{kg} = \Phi_g^{mass} \otimes [M_{gg} \Phi_g^{mass}]$$

where  $\Phi_g^{mass}$  represents the mass-normalized eigenvectors so that the total grid point kinetic energy is scaled to be unity. Note that the operator  $\otimes$  indicates term-wise matrix multiplication.

5. The grid point kinetic energy output has limited meaning for a coupled mass formulation. Since this mass formulation produces a coupling of mass across grid points, the sharing of kinetic energy among grid points can occur. In general, this obscures the meaning of the computation as a means of identifying important model parameters to control modal behavior.

**GPRSORT****Composites Ply Results Sorted Output**

Request sorted output of composites ply results (stress, strain, and failure indices) by global ply ID for a given element set.

Format:

$$GPRSORT = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

Examples:

GPRSORT=ALL  
GPRSORT=22

| Descriptor | Meaning  |
|------------|--|
| ALL        | All composite elements referencing a PCOMPG property entry type. See Remarks 1. and 2. |
| n          | Set identification number of a previously appearing SET command.                       |

Remarks:

1. Composite element output will be sorted by global ply ID and element ID. Note that this sorted output is only available for composite elements referencing a PCOMPG property entry. Global ply IDs can only be specified on the PCOMPG entry.
2. Composite elements referencing the PCOMP property entry will be excluded from the sorted output.

**GPSDCON****Grid Point Stress Discontinuity Output Request**

Requests mesh stress discontinuities based on grid point stresses.

**Format:**

$$GPSDCON \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

**Examples:**

GPSDCON=ALL

GPSDCON=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| ALL  | Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.   |
| n    | Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request (Integer > 0). |
| NONE | No grid point stress discontinuity output.  |

**Remarks:**

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS and STRFIELD commands must be present for printed output.

**GPSTRAIN**

## Grid Point Strain Output Request for Printing Only

Requests grid point strains for printing only.

**Format:**

$$GPSTRAIN \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

GPSTRAIN=ALL  
GPSTRAIN=19

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |  |
|------|--|
| ALL  | Grid point strain requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.   |
| n    | Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point strain output request (Integer > 0). |
| NONE | No grid point strain output.   |

**Remarks:**

- For statics, normal modes, and transient analysis, output will be presented for each surface or volume as a tabular listing of grid point strains for each load, eigenvalue, and time step. (See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.)
- Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
- Element strain output (STRAIN) must be requested for elements referenced on SURFACE and VOLUME commands.

4. In nonlinear transient analysis, grid point strains are computed only if parameter LGDISP is -1, which is the default, and only for elements with linear material properties.
5. For the postprocessing of grid point strains using the .xdb file or the computation of mesh strain discontinuities, the STRFIELD command must also be specified.

**GPSTRESS**

## Grid Point Stress Output Request for Printing Only

Requests grid point stresses for printing only.

**Format:**

$$GPSTRESS \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

GPSTRESS=ALL  
GPSTRESS=19

| Descriptor          | Meaning             |                   |                      |
|---------------------|---------------------|-------------------|----------------------|
|                     | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or<br>(blank) | X                   |                   | X*                   |
| PUNCH               |                     | X                 | X*                   |
| PLOT                |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |  |
|------|--|
| ALL  | Grid point stress requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.   |
| n    | Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point stress output request (Integer > 0). |
| NONE | No grid point stress output.   |

**Remarks:**

- For statics, normal modes, and transient analysis, output will be presented for each surface or volume as a tabular listing of grid point stresses for each load, eigenvalue, and timestep. (See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.)
- Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands.
4. For the postprocessing of grid point stresses using the .xdb file or the computation of mesh stress discontinuities, the STRFIELD command must also be specified.
5. Grid point stress is not output for midside nodes.

**GROUNDCHECK****Rigid Body Motion Grounding Check**

Perform grounding check analysis on the stiffness matrix to expose unintentional constraints by moving the model rigidly.

Format:

$$\text{GROUNDCHECK} \left[ \begin{array}{l} \left( \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right), \text{PUNCH}, \text{SET} = \left\{ \begin{array}{l} \left\{ \begin{array}{l} \text{G}, \text{N}, \text{N} + \text{AUTOSPC}, \text{F}, \text{A} \\ \text{ALL} \end{array} \right\} \\ \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\} \end{array} \right\} \\ \text{GRID} = \text{gid}, \text{THRESH} = \text{e}, \text{DATAREC} = \left[ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], (\text{RTHRESH} = \text{r}) \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

Examples:

GROUNDCHECK=YES

GROUNDCHECK (GRID=12, SET=(G, N, A), THRESH=1.E-5, DATAREC=YES) =YES

| Descriptor | Meaning   |
|------------|---|
| PRINT      | Write output to the print file. (Default)   |
| NOPRINT    | Do not write output to the print file.  |
| PUNCH      | Write output to the punch file.   |
| SET        | Selects degree-of-freedom set(s) (Default: SET=G).  |
| gid        | Reference grid point for the calculation of the rigid body motion.  |
| e          | Maximum strain energy which passes the check. The default value is computed by dividing the largest term in the stiffness matrix by 1.E10.    |
| DATAREC    | Requests data recovery of grounding forces (Default: DATAREC=NO).   |
| r          | Grounding forces which are larger than r percent of the largest grounding force will be printed if DATAREC=YES (Default = .10; 0. < r < 1.0). |

Remarks:

1. GROUNDCHECK must be specified above the subcase level.
2. SET=N+AUTOSPC (check the N-set stiffness, including the effect of PARAM,AUTOSPC) uses the stiffness matrix for the n-set, with the rows corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.
3. If DATAREC=YES, GROUNDCHECK FORCES will be printed in the displacement coordinate system of the associated GRID points.
4. For CBEAM/CBEAM3, it is recommended to use SPOINT ID for warping DOFs. Use of GRID ID for warping DOFs may cause one or more directions to fail the rigid body check.

5. If Lagrange multipliers are present via RIGID=LAGRAN or LGELIM then for degree-of-freedom sets N, N+AUTOSPC, F, and A the check will be performed on degree-of-freedom sets NL, NL+AUTOSPC, FL, and AL. The output will also be labeled accordingly.

## GUST

### Aerodynamic Gust Load Requests

Selects the gust field in an aeroelastic response problem.

**Format:**

GUST=n

**Example:**

GUST=73

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a GUST Bulk Data entry (Integer > 0). |

**Remark:**

1. The choice of transient or frequency response GUST depends upon the type of TLOAD or RLOAD referenced on the selected GUST entry.

**GVECTOR****Eigenvector Output Request in SOLs 200 and 400**

Requests the form and type of eigenvector output in SOLs 200 and 400. For other solutions use **DISPLACEMENT (Case)**

Format:

$$GVECTOR(\begin{bmatrix} PRINT & PUNCH \\ & PLOT \end{bmatrix}) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

**GVECTOR=ALL**  
**GVECTOR (PUNCH) =NONE**

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |  |
|------|--|
| ALL  | Eigenvectors for all points will be output.  |
| NONE | Eigenvectors for no points will be output.   |
| n    | Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0). |

Remarks:

1. GVECTOR=NONE overrides an overall output request.
2. Output will be presented as a tabular listing of grid points for each eigenvector.
3. GVECTOR is intended to be specified in subcases with ANALYSIS=MFREQ, MTRAN, MCEIG, FLUTTER, and SAERO.

## HADAPT

### Mesh Adaptivity Activation and Control

Specifies Mesh adaptivity control parameters in SOLs 101 and 400.

**Format:**

HADAPT=N

**Example:**

HADAPT=1

| Descriptor | Meaning  |
|------------|--|
| N          | Identification number for a HADAPTL Bulk Data entry (Integer > 0). |

**Remarks:**

1. The HADAPT command can be used only in SOL 101 or SOL 400 with ANALYSIS=STATICS. HADPAT cannot be used in Linear Contact Analyses.
2. In SOL 101, with NO SUPERELEMENTS, the HADAPT command may appear either above all SUBCASEs or within specific SUBCASEs. In the last scenario, only stresses on the solution corresponding to the specific SUBCASE where the HADAPT command has been placed will be used to compute error indicators should the user requests an error indicator based refinement criterion (see Bulk Data entries, [HADACRI, 2123](#) and [HADAPTL, 2126](#)). If superelements are present, then the HADAPT command should appear only in Residual Superelement (SE0) SUBCASE structure.
3. In SOL 400, the HADAPT command can only be placed on a Linear static structural analysis SUBCASE (ANALYSIS=STATICS) either above all STEPS or within each single STEP. All STEPs must be Linear Static structural STEPS (ANALYSIS=STATICS). In other words an adaptive meshing linear analysis cannot be chained with any other analysis type.
4. HADAPT remeshing should only be performed on lower order elements.

**HARMONICS****Harmonic Analysis or Printout Control**

Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.

**Format for Axisymmetric Problems:**

$$HARMONICS = \left\{ \begin{array}{l} ALL \\ NONE \\ h \end{array} \right\}$$

**Format for Cyclic Symmetric Problems:**

$$HARMONICS = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

**Examples:**

HARMONICS=ALL

HARMONICS=32

| Describer | Meaning   |
|-----------|---|
| ALL       | All harmonics will be output in the case of axisymmetric shell or axisymmetric fluid problems. All harmonics will be used for analysis in cyclic symmetry problems. |
| NONE      | No harmonics will be output. This option is not available for use in cyclic symmetry problems.  |
| h         | Available harmonics up to and including harmonic h will be output in the case of axisymmetric shell or axisymmetric fluid problems (Integer $\geq 0$ ).             |
| n         | Harmonics specified in SET n will be used for analysis in cyclic symmetry problems (Integer $> 0$ ).  |

**Remarks:**

1. If no HARMONICS command is present in the Case Control Section for axisymmetric shell or fluid problems, printed output is prepared only for the zero harmonic.
2. This command must be present in cyclic symmetry problems with HARMONICS=ALL or n; otherwise, the program will abort without performing any analysis.
3. In cyclic symmetry analysis, n must be defined as a set of integers on a SET command.

**HDOT****Heat Transfer Rate of Change of Enthalpy Output Request**

Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).

**Format:**

$$\text{HDOT}\left[\left(\begin{matrix} \text{[SORT1]} \\ \text{[SORT2]} \end{matrix}\right), \left(\begin{matrix} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{matrix}\right)\right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

HDOT=5

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of grid points for each time. |                   |                      |
| SORT2            | Output will be presented as a tabular listing of time for each grid point.  |                   |                      |
| PRINT or (blank) | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| X                |   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| ALL  | Rate of change of enthalpy for all points will be output.   |
| NONE | Rate of change of enthalpy for no points will be output.  |
| n    | Set identification of previously appearing SET command. Only rates of change of enthalpy for points with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remark:**

1. HDOT=NONE is used to override a previous HDOT=n or HDOT=ALL command.

## HISTOGRAM

### Fatigue Histogram/Time History Output Request

Requests resultant stress/strain time history and/or rainflow cycle histogram output from entities defined by a FATIGUE case control output request for SOLs 101, 103, and 112 only.

Format:

$$\text{HISTOGRAM} \left[ \begin{bmatrix} \text{BOTH} \\ \text{HIST} \\ \text{TIME} \end{bmatrix}, \text{FORMAT} = [\text{CODE}] \right] = \begin{cases} \text{ALL} \\ n \\ \text{NONE} \end{cases}$$

Examples:

```
SET 99 = 100,101
HISTOGRAM (TIME,FORMAT=ASCII) = 99
```

| Descriptor | Meaning   |
|------------|---|
| BOTH       | Both time histories and histograms are output (Default).  |
| HIST       | Only histograms are output.   |
| TIME       | Only time histories are output.   |
| FORMAT     | Specifies the output file format of the time history and/or histogram files. Valid values of CODE are ASCII or BINARY. (Default: FORMAT=ASCII).                 |
| ALL        | Time histories and/or histograms of all entities associated with a FATIGUE case control output request are output. See Remarks below.                           |
| n          | Time histories and/or histograms of only entities defined in a SET of ID n are output. The entities must also be part of a FATIGUE case control output request. |
| NONE       | No histogram or time history output is processed.   |

Remarks:

1. If no FATIGUE case control is defined, this output request is ignored.
2. This output request has the potential to create many thousands of files if the ALL option is used. A separate file for each time history and/or histogram for each entity is created. It is recommended to specify a SET to limit the output.
3. If FORMAT=BINARY, file formats are DAC for time histories and S3M for histograms.  
If FORMAT=ASCII, both time histories and histograms are output in CSV format.
4. File names produced by this option are cryptic and increment by the number of time histories or histograms being processed. You must open each file to check for which element, node, and layer it is associated with.

5. This option is not supported for duty cycle analysis and will be ignored with no output generated. Specifically this occurs when the FTGSEQ bulk data entry references more than one FTGEVNT or more than one repeat of a single FTGEVNT entry.

**HOUTPUT**

## Harmonic Output Request in Cyclic Symmetry Problems

Requests harmonic output in cyclic symmetry problems.

Format:

$$\text{HOUTPUT}[(\mathbf{C}, \mathbf{S}, \mathbf{C}^*, \mathbf{S}^*)] = \left\{ \begin{array}{l} \text{ALL} \\ K \end{array} \right\}$$

Examples:

HOUTPUT=ALL

HOUTPUT(C, S)=5

| Descriptor   | Meaning  |
|--------------|--|
| C, S, C*, S* | Harmonic coefficients. See Remark 4.                             |
| ALL          | All harmonics will be output.                                    |
| k            | Set identification number of harmonics for output (Integer > 0). |

Remarks:

1. Set k must be defined on a SET command, and output will be computed for all available harmonics in SET k.
2. HOUTPUT=ALL requests output for all harmonics specified on the HARMONICS command.
3. Either the HOUTPUT or NOUTPUT command is required to obtain data recovery in cyclic symmetry analysis.
4. C and S correspond to the cosine and sine coefficients when the STYPE field is ROT or AXI on the CYSYM Bulk Data entry.  
C, S, C\*, and S\* correspond to the cosine symmetric, sine symmetric, cosine antisymmetric, and sine antisymmetric coefficients, respectively, when the STYPE field is DIH on the CYSYM Bulk Data entry.

**HTFLOW****Elemental Heat Flow Output Request**

Requests heat flow output at selected structural elements.

**Format:**

$$\text{HTFLOW} \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{NOPRINT} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$$

**Example:**

HTFLOW = ALL  
HTFLOW = 15

| Descriptor | Meaning  |
|------------|--|
| PRINT      | The printer will be the output medium.   |
| NOPRINT    | Generate, but do not print out, the output.  |
| PUNCH      | The punch file will be the output medium.  |
| ALL        | Heat flow for all structural elements will be output.  |
| n          | Set identification of previously appearing SET command. Only structural elements with identification numbers that appear on this SET command will be included in the heat flow output (Integer > 0). |

**Remarks:**

1. Elemental heat flow output is available for steady state thermal analysis (SOL 101 and SOL 153) and transient thermal analysis (SOL 159).
2. Heat flow is computed from the applied heat loads and the effect of convection and radiation heat transfer on boundary elements (CHBDYE, CHBDYG, and CHBDYP).
3. See Remarks 6-8. of the descriptions of CHBDYE Bulk Data for the side conventions of solid elements, shell elements, and line elements.

**IC****Transient Analysis Initial Condition Set Selection**

Selects the initial conditions for transient analysis (SOLs 109, 112, 129, 159, 400 and 600).

**Format:**

$$IC \begin{bmatrix} PHYSICAL \\ MODAL \\ STATSUB[,DIFFK] \end{bmatrix} = n$$

**Examples:**

```
IC = 10
IC(PHYSICAL) = 100
IC(MODAL) = 200
IC(STATSUB) = 1000
IC(STATSUB,DIFFK) = 2000
```

| Descriptor | Meaning   |
|------------|---|
| PHYSICAL   | The TIC Bulk Data entries selected by set n define initial conditions for coordinates involving grid, scalar, and extra points (Default).   |
| MODAL      | The TIC Bulk Data entries selected by set n define initial conditions for modal coordinates and extra points. See Remark 3.   |
| STATSUB    | Use the solution of the static analysis subcase n as the initial condition. See Remark 4.   |
| DIFFK      | Include the effects of differential stiffness in the solution. See Remarks 4. and 5.  |
| n          | For the PHYSICAL (Default) and MODAL options, n is the set identification number of TIC Bulk Data entries for structural analysis (SOLs 109, 112, 129, and 600) or TEMP and TEMPD entries for heat transfer analysis (SOLs 159, 400 and 600). For the STATSUB option, n is the ID of a static analysis subcase (Integer > 0). |

**Remarks:**

1. For structural analysis, TIC entries will not be used (therefore, no initial conditions) unless selected in the Case Control Section.
2. Only the PHYSICAL option (Default) may be specified in heat transfer analysis (SOLs 159, 400 and 600). The initial temperature of a point should be equal to its boundary temperature at time = 0. For SOL 400 transient thermal analysis, the boundary temperature at time = 0 takes precedence over the initial temperature if both temperatures are not identical.
3. IC(MODAL) may be specified only in modal transient analysis (SOL 112).
4. IC(STATSUB) and IC(STATSUB,DIFFK) may not both be specified in the same execution.
5. The DIFFK keyword is meaningful only when used in conjunction with the STATSUB keyword.
6. The following examples illustrate the usage of the various options of the IC Case Control command.

Example (a)

```
$ SPECIFY INITIAL CONDITIONS FOR PHYSICAL COORDINATES
$ IN SOL 109 OR SOL 112
IC(PHYSICAL) = 100
```

or

```
IC = 100
```

Example (b)

```
$ SPECIFY INITIAL CONDITIONS FOR MODAL COORDINATES
$ IN SOL 112
IC(MODAL) = 200
```

Example (c)

```
$ SPECIFY STATIC SOLUTION AS INITIAL CONDITION
$ IN SOL 109 OR SOL 112
$ (DIFFERENTIAL STIFFNESS EFFECT NOT INCLUDED)
SUBCASE 10 $ STATIC ANALYSIS
LOAD = 100
SUBCASE 20 $ TRANSIENT ANALYSIS
IC(STATSUB) = 10 $ POINTS TO STATIC ANALYSIS SUBCASE ID
```

Example (d)

```
$ SPECIFY STATIC SOLUTION AS INITIAL CONDITION
$ IN SOL 109 OR SOL 112
$ (DIFFERENTIAL STIFFNESS EFFECT INCLUDED)
SUBCASE 100 $ STATIC ANALYSIS
LOAD = 1000
SUBCASE 200 $ TRANSIENT ANALYSIS
IC(STATSUB, DIFFK) = 100 $ POINTS TO STATIC ANALYSIS SUBCASE ID
```

**ICF****Inter-Component Forces (ICF) Output Request**

Requests the form and type of inter-component force (ICF) vector output from an FBA job.

**Format:**

$$ICF \left( \begin{bmatrix} PHYSICAL \\ MODAL \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix} \right) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

ICF = 5

ICF (SORT2, PRINT, PUNCH, IMAG) = ALL

ICF (SORT2) = 20

ICF = ALL

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of grid points for each frequency. See Remarks 1. through 4. |
| SORT2      | Output will be presented as a tabular listing of frequency for each grid point. See Remarks 1., 3. and 5.  |

|                     | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|---------------------|---------------------|-------------------|----------------------|
| PRINT or<br>(blank) | X                   |                   | X*                   |
| PUNCH               |                     | X                 | X*                   |
| PLOT                |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |
| ALL          | ICFs for all points will be output. See Remarks 1. through 5.  |

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification of a previously appearing SET command. Only ICFs for points with identification numbers that appear on this SET command will be output (Integer > 0). See Remarks 1. through 6. |
| NONE       | ICFs for no points will be output.   |

**Remarks:**

1. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In SORT1 format, ICFs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
3. ICFs are defined only for the connection points of FRF components in an FBA job and are regarded as null for non-connection points of FRF components. Therefore, ICF output is meaningful only for the connection points of FRF components in an FBA job.
4. In SORT1 format, ICF output is ignored for non-connection points and will be output for the requested connection points of FRF components only if their ICFs are non-null.
5. In SORT2 format, ICF output is honored for all requested connection and non-connection points of FRF components even if their ICFs are null.
6. ICF output request is ignored for undefined points.

## IMPERFECT

### Select Geometric Imperfection

Select geometric imperfection cases in SOL 400.

Format:

IMPERFECT = n

Example:

IMPERFECT = 5

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

|   |   |
|---|---|
| n | Identification number of an IMPGEOM or IMPCASE bulk data entry. |
|---|---|

Remark:

1. This command must be above all subcases.

## INCLUDE

### Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

INCLUDE 'filename'

**Example:**

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYCASE.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
INCLUDE 'MYCASE.DATA'
BEGIN BULK
ENDDATA
```

| Descriptor | Meaning  |
|------------|--|
| filename   | Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (''). |

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file:

/dir123/dir456/dir789/filename.dat

may be included with the following input:

```
INCLUDE '/dir123
          /dir456
          /dir789/filename.dat'
```

3. See the [MSC Nastran 2020 Installation and Operations Guide](#) for more examples.

**INTENSITY****Acoustic Intensity Output Request**

Requests output of acoustic intensity on wetted surface. SOLs 108 and 111 only.

Format:

$$\text{INTENSITY} \left[ \left( \begin{matrix} \text{[SORT1]} \\ \text{[SORT2]} \end{matrix}, \begin{matrix} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{matrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as tabular listing of grid points for each excitation frequency (Default). |
| SORT2      | Output will be presented as tabular listing of excitation frequencies for each grid point.          |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| ALL  | Intensities will be computed for all grid points of the wetted surface.   |
| n    | Set identification of a previously defined set of grid points. Intensities will be computed for the grid points in this set only. |
| NONE | Acoustic intensities will not be processed.   |

Remarks:

1. INTENSITY = NONE overrides an overall request.
2. This Case Control command can be used in SOL 108 and SOL 111 only.

**IRLOAD**

## Nonlinear Inertia Relief Selection (SOL 400 only)

Selects nonlinear inertia relief set for SOL 400

Format:  $IRLOAD = \left\{ \begin{array}{l} QLINEAR \\ \boxed{NONE} \end{array} \right\}$

Example:

IRLOAD=QLINEAR

| Descriptor | Meaning  |
|------------|--|
| QLINEAR    | Inertia Load Calculation with small displacement (Quasi-Linear) is activated in SOL 400. |
| NONE       | No Inertia Relief (Default)  |

Remark:

1. This command is active only in SOL 400 with ANALYSIS=NLSSTATIC and its use requires a set of STATIC supports that constrain all six rigid body motions.
2. IRLOAD=QLINEAR, which has to be applied above to all SUBCASES, is a global case control command and activates the inertia load calculations in SOL 400 for all applied static loads. In nonlinear static analyses (ANALYSIS=NLSSTAT), it also activates the inertia relief analysis with small displacement. When IRLOAD=QLINEAR with large displacement (PARAM,LGDISP,1), a fatal error message will be issued. Also superelements in conjunction with IRLOAD=QLINEAR will cause a fatal error.  
IRLOAD=NONE (default) deactivates the inertia load calculations.
3. IRLOAD=QLINEAR is ignored by perturbation analyses in SOL 400.

**K2GG****Direct Input Stiffness Matrix Selection**

Selects direct input stiffness matrix or matrices.

**Format:**

K2GG=name

**Example:**

```
K2GG = KDMIG
K2GG = KDMIG1, KDMIG2, KDMIG3
K2GG = 1.25*KDMIG1, 1.0*KDMIG2, 0.82*KDMIG3
SET 100 = K2, K3, K4
K2GG = 100
```

| Descriptor | Meaning   |
|------------|---|
| name       | Name of a $[K_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 6. (Character). |

**Remarks:**

1. DMIG matrices will not be used unless selected.
2. Terms are added to the stiffness matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
4. A scale factor may be applied to this input using the PARAM, CK2 entry. See [Parameters, 783](#).
5. The matrices are additive if multiple matrices are referenced on the K2GG command.
6. The formats of the name list:
  - a. Names without factor.  
Names separated by comma or blank.
  - b. Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
7. If there are multiple subcases in the analysis, the K2GG command must appear above the first subcase or in the first subcase. K2GG requests in the second and subsequent subcases will be ignored. For superelements, it should occur in the first subcase for the appropriate superelement.

**K2PP****Direct Input Stiffness Matrix Selection**

Selects direct input stiffness matrix or matrices, which are not included in normal modes.

**Format:**

K2PP=name

**Example:**

K2PP = KDMIG

K2PP = KDMIG1, KDMIG2, KDMIG3

K2PP = 5.06\*KDMIG1, 1.0\*KDMIG2, 0.85\*KDMIG3

K2PP = (1.25, 0.5) \*KDMIG1, (1.0,0.0) \*KDMIG2, (0.82, -2.2) \*KDMIG3

| Descriptor | Meaning   |
|------------|---|
| name       | Name of a $[K_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors. See Remark 6. |

**Remarks:**

1. DMIG and DMIAX entries will not be used unless selected by the K2PP command.
2. The matrix must be square or symmetric, and field 4 on the DMIG,name Bulk Data entry must contain a 1 or 6.
3. It is recommended that PARAM,AUTOSPC,NO be specified. See the [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
4. K2PP matrices are used only in dynamic response problems. They are not used in normal modes.
5. The matrices are additive if multiple matrices are referenced on the K2PP Case Control command.
6. The formats of the name list:
  - a. Names without factor  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers separated by a comma, within parentheses, as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be paired with a factor including 1.0 for real and (1.0, 0.0) for complex.

**K42GG****Direct Input Stiffness Element Damping Selection**

Selects direct input structural element damping matrix or matrices.

**Format:**

K42GG=name

**Example:**

K42GG = KDMIG

K42GG = KDMIG1, KDMIG2, KDMIG3

K42GG = 2.03\*KDMIG1, 0.84\*KDMIG2

| Descriptor | Meaning   |
|------------|---|
| name       | Name of a $[K4_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 4. |

**Remarks:**

1. DMIG matrices will not be used unless selected.
2. Terms are added to the structural element damping matrix before any constraints are applied.
3. The matrix must be symmetric, and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
4. The formats of the name list:
  - a. Names without factor.  
Names separated by comma or blank.
  - b. Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
5. If there are multiple subcases in the analysis, the K42GG command must appear above the first subcase or in the first subcase. K42GG requests in the second and subsequent subcases will be ignored. For superelements, it should occur in the first subcase for the appropriate superelement.

## LABEL

### Output Label

Defines a character string that will appear on the third heading line of each page of printer output.

**Format:**

LABEL=label

**Example:**

LABEL=DEMONSTRATION PROBLEM

| Descriptor | Meaning               |
|------------|-----------------------|
| label      | Any character string. |

**Remarks:**

1. LABEL appearing at the subcase level will label output for that subcase only.
2. LABEL appearing before all subcases will label any outputs that are not subcase-dependent.
3. If no LABEL command is supplied, the label line will be blank.
4. LABEL information is also placed on plotter output, as applicable. Only the first 65 characters will appear.

**LINE** Maximum Lines Per Printed Page

Defines the maximum number of output lines per printed page.

Format:

LINE=n

Example:

LINE=35

| Descriptor | Meaning  |
|------------|--|
| n          | Maximum number of output lines per page (Integer > 0; Default = 50). |

Remarks:

1. For 11 inch paper, 50 lines per page is recommended; for 8-1/2 inch paper, 35 lines per page is recommended.
2. The NASTRAN statement keyword NLINES may also be used to set this value. See the [Executing MSC Nastran, 1](#).

**LOAD****External Static Load Set Selection**

Selects an external static load set.

**Format:**

LOAD=n

**Example:**

LOAD=15

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one ACCEL, ACCEL1, FORCE, FORCE1, FORCE2, FORCEAX, GRAV, LOAD, MOMAX, MOMENT, MOMENT1, MOMENT2, MPCD, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADB3, PLOADX, QVOL, QVECT, QHBDY, QBDY1, QBDY2, QBDY3, PRESAX, RFORCE, SPCD, or SLOAD entry (Integer > 0). |

**Remarks:**

1. A GRAV entry cannot have the same set identification number as any of the other loading entry types.  
Apply a gravity load along with other static loads, a LOAD Bulk Data entry must be used.
2. LOAD is only applicable in linear and nonlinear statics, inertia relief, differential stiffness, buckling, and heat transfer analyses.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) loads.
4. Static, thermal, and element deformation loads should have unique set identification numbers.

## LOADNAME

Name to be Associated with a SUBCASE in PAA

Provides a name to be associated with a loading condition.

Format:

LOADNAME = load\_name

Example:

LOADNAME = GRAVITY LOAD ON OUTBOARD NACELLEL

| Describer | Meaning   |
|-----------|---|
| Load_name | Character string (up to 64 characters long) providing a user-defined name for a loading in PAA. |

Remark:

1. This command is used only in PAA and provides a user-defined name for a loading condition.
2. LOADNAMEs may be used to combine loads in a COMBINE or SOLVE PAA run using a LOADCNAM Bulk Data entry.
3. The LOADNAME command is ignored in non-PAA applications

**LOADSET****Static Load Set Selection**

Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.

**Format:**

LOADSET=n

**Example:**

LOADSET=100

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification number of at least one LSEQ Bulk Data entry. (Integer > 0) |

**Remarks:**

1. When used in superelement analysis, this command must be used for all superelements. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
2. When the LOADSET command is used in superelement statics, the residual structure should have as many loading conditions as the number of unique EXCITEID sets defined on all LSEQ entries. The subcases after the first should contain only SUBTITLE and LABEL information, and residual structure output requests. SUBTITLE and LABEL information for all superelements will be obtained from the residual structure.
3. When multiple subcases are specified in the dynamic solution sequences (SOLs 108, 109, 111, 112, 118, 146), the LOADSET must appear in the first subcase or above all subcases. In SOL 200 with ANALYSIS=DFREQ, MFREQ, or MTRAN, a different LOADSET may be specified in the first subcase pertaining to each ANALYSIS command.
4. In SOL 101, the design sensitivity output will identify all expanded subcases by a sequence of unique integers beginning with n.
5. In the nonlinear static solution sequences (SOLs 106 and 153), the LOADSET must appear above all subcases and only one LOADSET may be specified.
6. Only one LOADSET command is allowed per superelement and it must be specified in the superelement's first subcase.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of a LOADSET Case Control command, all static loads whose load set IDs match the EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE Bulk Data entries are automatically processed.

8. . If you use the LOADSET command in a static analysis, the loading matrix will be generated corresponding to, and in ascending order of, the EXCITEID's on the referenced LSEQ entries. Any LOAD commands within subcases will be ignored.
9. Use of the LOADSET/LSEQ should be avoided as it is an obsolete entry and is never needed and is only documented for legacy code. Some new features such as SUBSTEP do not support it and will issue a fatal message.

**M2GG****Direct Input Mass Matrix Selection**

Selects direct input mass matrix or matrices.

**Format:**

M2GG=name

**Example:**

```
M2GG = MDMIG
M2GG = MDMIG1, MDMIG2, MDMIG3
M2GG = 1.25*MDMIG1, 1.0*MDMIG2, 0.82*MDMIG3
SET 200 = M1, M2
M2GG = 200
```

| Descriptor | Meaning  |
|------------|--|
| name       | Name of a $[M_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors see Remark 6. (Character). |

**Remarks:**

1. DMIG matrices will not be used unless selected.
2. Terms are added to the mass matrix before any constraints are applied.
3. The matrix must be symmetric, and field 4 on the DMIG, name entry must contain a 6.
4. M2GG input is not affected by PARAM,WTMASS. M2GG input must either be in consistent mass units or PARAM,CM2 may be used.
5. The matrices are additive if multiple matrices are referenced on the M2GG command.
6. The formats of the name list:
  - a. Names without factor.  
Names separated by comma or blank.
  - b. Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
7. If there are multiple subcases in the analysis, the M2GG command must appear above the first subcase or in the first subcase. M2GG requests in the second and subsequent subcases will be ignored. For superelements, it should occur in the first subcase for the appropriate superelement.

**M2PP****Direct Input Mass Matrix Selection**

Selects direct input mass matrix or matrices, which are not included in normal modes.

**Format:**

M2PP=name

**Example:**

M2PP = MDMIG

M2PP = MDMIG1, MDMIG2, MDMIG3

M2PP = 5.06\*MDMIG1, 1.0\*MDMIG2, 0.85\*MDMIG3

M2PP = (1.25, 0.5) \*MDMIG1, (1.0, 0.0) \*MDMIG2, (0.82, -2.2) \*MDMIG3

| Descriptor | Meaning  |
|------------|--|
| name       | Name of a $[M_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors, see Remark 7. (Character). |

**Remarks:**

1. DMIG and DMIAX entries will not be used unless selected by the M2PP input.
2. M2PP input is not affected by PARAM,WTMASS. M2PP input must be in consistent mass units.
3. The matrix must be square or symmetric, and field 4 on the DMIG, name entry must contain a 1 or 6.
4. It is recommended that PARAM,AUTOSPC,NO be specified. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
5. M2PP matrices are used only in dynamic response problems. They are not used in normal modes problems.
6. The matrices are additive if multiple matrices are referenced on the M2PP command.
7. The formats of the name list:
  - a. Names without factor  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factors followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parenthesis as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

**MASSET****Mass Combination Selection**

Selection of a mass combination in multiple mass combination analysis (MMC). This case control command can be used in various solutions sequences SOLs 101, 103, 107-112, 144-146, 200 and 400.

**Format:**

MASSET = n

**Example:**

MASSET = 11

| Descriptor | Meaning   |
|------------|---|
| n          | Set ID of a MASSET bulk data entry (Integer > 0). |

**Remarks:**

1. Using a particular mass combination in analysis only requires referencing the mass combinations ID in MASSET case control at the subcase level (or at step level in Sol 400). If no MASSET is defined in the subcase then (like traditional non MMC analysis) the base mass is used as the mass for that subcase. Example for selecting mass combinations at the subcase level:

```
SUBCASE 101
...
MASSET = 11 $
...
SUBCASE 102
...
MASSET = 12 $
```

2. The MASSET bulk data entry defines a linear combination of incremental/base mass cases to form the subcase-dependent mass. The bulk data for the incremental mass case is preceded by "BEGIN MASSID=yyy", where yyy > 0 is the integer ID of the mass increment case. The only data that will be used from the incremental mass bulk data section is element connections and element mass properties including materials (Parameters and grid data included in the incremental mass section will be ignored). The incremental mass will be computed based on the grid data supplied in the base mass bulk data section and the element properties supplied in the incremental mass section. Typically the user will construct the incremental mass by adding concentrated masses (e.g. CONM2, CMASS1 etc.) or by changing the material or element properties of the set of elements defined in incremental mass bulk data section.

```
BEGIN massid=101 label='mass increment using conm2s'
BEGIN massid=102 label='mass increment using materials'
```

3. If Part SE are used in the model, mass increment related to these SEs can be specified in a separate bulk data section, "BEGIN SUPER=zzz MASSID=yyy", where zzz > 0 is the Superelement ID and yyy is the mass increment ID.

```
BEGIN SUPER=2 massid=101 label='mass increment using conm2s'
```

**MASTER****Redefine the MASTER Subcase**

Allows the redefinition of a MASTER subcase.

**Format:**

SUBCASE n  
MASTER

**Example:**

SUBCASE 10  
MASTER

| Descriptor | Meaning |
|------------|---------|
| n          |         |

**Remarks:**

1. All commands in a MASTER subcase apply to the following subcases until a new MASTER subcase is defined.
2. Suppose that superelement 10 has SPC set 10, MPC set 10, and LOAD sets 101 and 102. Suppose also that superelement 20 has SPC set 20, MPC set 20, and LOAD sets 201 and 202. Then the following Case Control setup specifies the required subcase structure:

```
TITLE = MY MODEL
DISP = ALL
SEALL = ALL
SUBCASE 101
MASTER
SPC = 10
MPC = 10
SUPER = 10, 1
LOAD = 101
LABEL = SUPER 10
ESE = ALL
SUBCASE 102
LOAD = 102
SUPER = 10, 2
SUBCASE 201
MASTER
SPC = 20
MPC = 20
SUPER = 20, 1
LOAD = 201
LABEL = SUPER 20
SUBCASE 202
LOAD = 202
SUPER = 20, 2
```

3. MASTER may also be used advantageously in multiple boundary condition Case Control setups. Suppose that constraint sets 10 and 20 are to be solved with three loading conditions each: 1, 2, and 3; and 4, 5, and 6, respectively. The following Case Control Section may be used:

```
TITLE = MULTIPLE BOUNDARY CONDITIONS
DISP = ALL
SYM 1
MASTER
    SPC = 10
    LOAD = 1
SYM 2
    LOAD = 2
SYM 3
    LOAD = 3
SYM 4
    MASTER
    SPC = 20
    LOAD = 4
SYM 5
    LOAD = 5
SYM 6
    LOAD = 6
SYMCOM 10
    SYMSEQ = 1., 1., 1., -1., -1., -1.
SYMCOM 20
    SYMSEQ = -1., -1., -1., 1., 1., 1.
```

4. The MASTER command must appear immediately after a SUBCASE or SYM command.

**MAXLINES****Maximum Number of Output Lines**

Sets the maximum number of output lines.

Format:

MAXLINES=n

Example:

MAXLINES=150000

| Describer | Meaning  |
|-----------|--|
| n         | Maximum number of output lines allowed (Integer > 0; Default = 999999999). |

Remarks:

1. If MAXLINES is exceeded, the program will terminate.
2. MAXLINES does not override any system parameters such as those on job control language commands.
3. MAXLINES may also be specified on the NASTRAN statement with SYSTEM(14). See the [Executing MSC Nastran](#).
4. The code counts the number of pages and assumes that the number of lines output is the number of lines allowed per page, specified by the “LINES” command, times the number of pages.

**MAXMIN****MAXMIN Survey Output Request (old form)**

Specifies options for max/min surveys of certain output data associated with grid points.

**Format:**

$$\text{MAXMIN} \left( \begin{bmatrix} \text{MAX} \\ \text{BOTH} \\ \text{MIN} \\ \text{VMAG} \end{bmatrix}, = \text{num} \right), \left[ \text{CID} = \begin{cases} \text{GLOBAL} \\ \text{BASIC} \\ \text{cid} \end{cases} \right], \text{oplist}, \text{COMP} = \text{list} \right) = \begin{cases} \text{ALL} \\ n \\ \text{NONE} \end{cases}$$

**Example:**

MAXMIN (BOTH=10,CID=1000,DISP,COMP=T1/T3)=501

| Descriptor | Meaning   |
|------------|---|
| MAX        | Specifies that only maximum values are to be output. See Remark 1.  |
| MIN        | Specifies that only minimum values are to be output. See Remark 1.  |
| BOTH       | Specifies that both maximum and minimum values are to be output. See Remark 1.  |
| VMAG       | Specifies that vector magnitude resultants are to be output. See Remark 2.  |
| num        | Specifies the maximum number of values that will be output. See Remark 3.<br>(Integer > 0, Default = 5).  |
| CID        | Specifies the coordinate system frame of reference in which the max/min values will be output. See Remarks 1. and 3.  |
| GLOBAL     | Requests output in the global coordinate system frame of reference.   |
| BASIC      | Requests output in the basic coordinate system frame of reference.  |
| cid        | Requests output in the local coordinate system defined by cid (Integer > 0).  |
| oplist     | Specifies a list of one or more standard Case Control commands for which max/min results are to be produced. The list may include any of DISP,SPCF, OLOAD,MPCF,VELO,ACCE, or ALL. See Remark 6 (Character, no Default).   |
| list       | Specifies a list of grid point degree of freedom component directions that will be included in the max/min survey output. The components are separated by slashes and are selected from T1, T2, T3, R1, R2, and R3. See Remarks 4. and 5.<br>(Character, Default=/T1/T2/T3/R1/R2/R3). |
| ALL        | MAXMIN survey results for all points will be output.  |
| NONE       | MAXMIN survey results for no points will be output.   |
| n          | Set identification of a previously appearing SET command. The max/min results survey will be output only for the points specified SET n (Integer > 0).  |

**Remarks:**

1. The MAXMIN command produces an algebraically ascending sorted list of the output quantities specified for all of the points in the selected set. MAX refers to the largest magnitude positive values, while MIN refers to the largest magnitude negative values. The output format is similar to that of displacement output. All components will be output for a grid point, and the order of the grid points will be in sort on the particular component that was surveyed. The output title contains the identification number of the set of points participating in the max/min output, the coordinate system frame of reference, the number of MAX and MIN values output, and the component that was surveyed. When the output being surveyed is in the global output coordinate system reference frame, and BASIC or a local output coordinate system is specified as cid, both the sorted system output and the original reference system output are displayed for the grid point if these systems are different.
2. Vector magnitude results are computed for both translations and rotations and are displayed under the T1 and R1 column headings. The presence of the COMP keyword is ignored.
3. The default value of 5 generates a minimum of 10 output lines for the BOTH option. There will be 5 maximum values and 5 minimum values produced. In addition, if coordinate system are involved, both surveyed and original data will be output. This could result in as many as 10 more lines of output for each surveyed component.
4. Multiple MAXMIN commands may be specified for a subcase. This permits different output quantities to have different MAXMIN specification within a subcase. For example,

```
SET 501=1,3,5,7 THRU 99, 1001, 2001  
MAXMIN (DISP, COMP=T3) = 501  
MAXMIN (SPCF, COMP=T1/R3) = ALL
```

5. Scalar point output is included only if component T1 is included in the list.
6. MAXMIN output will only be generated for items in the oplist when there is an associated Case Control command present. For example, a DISP Case Control command must be present in order for the MAXMIN(DISP) = ALL command to produce output. Use of ALL keywords for the oplist requests MAXMIN output for all output commands acceptable to MAXMIN that are present in Case control Section.

**MAXMIN**

Requests Output of Maximums and Minimums in Data Recovery (new form)

Requests the output of maximums and minimums in data recovery.

**Format:**

$$\left[ \text{MAXMIN} \left( \begin{array}{l} \text{GRID} \\ \text{ELEM} \\ \text{BOTH} \end{array} \right), \left[ \begin{array}{l} \text{PRINT}, \text{PUNCH} \\ \text{NOPRINT} \end{array} \right], \left[ \begin{array}{l} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \left[ \begin{array}{l} \text{CENTER} \\ \text{CUBIC} \\ \text{SGAGE} \\ \text{CORNER or BILIN} \end{array} \right], \left[ \begin{array}{l} \text{STRCUR} \\ \text{FIBER} \end{array} \right] \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

MAXMIN (GRID) = 12  
 MAXMIN (ELEM) = ALL  
 MAXMIN = NONE

| Descriptor      | Meaning  |
|-----------------|--|
| GRID            | Indicates the request is applied to grid points.   |
| ELEM            | Indicates the request is applied to elements.  |
| BOTH            | Indicates the request is applied to both elements and grid points (Default).   |
| PRINT           | Compute and write the output to the .f06 file (Default).   |
| PUNCH           | Compute and write output to the punch file.  |
| NOPRINT         | Compute but do not write out the results.  |
| ALL             | Max/min results will be reported for all elements.   |
| n               | Set identification number. The referenced SET command defines a set of elements or grid points to be monitored.            |
| NONE            | Max/min results will not be reported.  |
| VONMISES        | von Mises stress/strains are output.   |
| MAXS or SHEAR   | Maximum shear stress/strains are output.   |
| STRCUR          | Strain at the reference plane and curvatures are output for plate elements.  |
| FIBER           | Stress/Strain at locations Z1, Z2 are computed for plate elements.   |
| CENTER          | Output CQUAD4 stress/strains at the center only.   |
| CORNER or BILIN | Output CQUAD4 element stress/strains at the center and grid points using strain gage approach with bilinear extrapolation. |
| SGAGE           | Output CQUAD4 element stress/strains at center and grid points using strain gage approach.                                 |
| CUBIC           | Output CQUAD4 element stress/strains at center and grid points using cubic bending correction.                             |

**Remarks:**

1. MAXMIN is not allowed in REPCASE, but is allowed in SUBCOM and SYMCOM.
2. The OTIME command may be used to limit the time span of monitoring.
3. No corresponding output request such as DISP, STRESS, etc., is required (Except for STRAIN). Also, SET=n may request more elements or grid points for monitoring than is requested by Outputs.

Currently for strain the output request STRAIN=n is required where n can reference any element. If strain at fibers is requested then, then STRAIN (FIBER) = n is required.

The output is comprised of two parts: (1) a summary of the maximum/minimum values and the times they occur, and (2) the associated output for all components of the element or grid. The first part is always output, and the second part is only output if the FULL descriptor is specified on the MAXMIN(DEF) Case Control command. See the next section for a description of the new Case Control commands. Here are sample Case Control commands for the output of maximum von Mises stresses using the BRIEF option:

```
MAXMIN (DEF) STRESS QUAD4 SMAX1 ABSOLUTE (5) BRIEF  
SET 100=4  
MAXMIN (ELEM) =4
```

4. See DISPLACEMENT, STRESS, or STRAIN Case Control commands for additional keyword implications.

**MAXMIN(DEF)**

Defines Parameters for Monitoring Maximums and Minimums (new form)

Defines parameters and output options for the monitoring of maximums and minimums in data recovery. MAXMIN(DEF) must be specified above all subcases. The MAXMIN(ELEM) and/or MAXMIN(GRID) Case Control command is then required to print the max/min results.

**Format for Grid Point Output:**

*MAXMIN(DEF){ DISP, VELO, ACCE, MPCF, SPCF, OLOAD }{ T1 T2 T3 R1 R2 R3 MAGT MAGR },*

$\left[ \begin{array}{c} ABSOLUTE(p) \ MINALG(q) \ MAXALG(r) \\ ALL(p) \end{array} \right] [RMS] \left[ \begin{array}{c} BRIEF \\ FULL \end{array} \right],$

$\left[ CID = \left\{ \begin{array}{c} GLOBAL \\ BASIC \\ cid \end{array} \right\} \right]$

**Format for element output:**

*MAXMIN(DEF) {*  $\left\{ \begin{array}{c} STRESS \\ STRAIN \\ FORCE \end{array} \right\}$  { *eltype1[, eltype2,... ]* } *{ comp1*  $\left[ \begin{array}{c} CENTER \\ ALL \\ GROUP \\ ENDS \end{array} \right]$  [, *comp2...*] } *}*

$\left[ \begin{array}{c} ABSOLUTE(p) \ MINALG(q) \ MAXALG(r) \\ ALL(p) \end{array} \right] [RMS] \left[ \begin{array}{c} BRIEF \\ FULL \end{array} \right]$

**Examples:**

MAXMIN(DEF) stress (quad4,smax1) MAXA=5  
 MAXMIN(DEF) disp T1 T2 T3 MAGT RMS FULL

| Descriptor                           | Meaning  |
|--------------------------------------|--|
| DISP, VELO, etc. (grid point output) | Type of result to be monitored.  |
| STRESS, etc. (element output)        |  |
| T1, T2, etc.                         | Name of the grid point component to be monitored.                            |
| MAGT                                 | Specifies the magnitude of the translational components are to be monitored. |
| MAGR                                 | Specifies the magnitude of the rotational components are to be monitored     |

| Descriptor          | Meaning  |
|---------------------|--|
| eltype1 eltype2 ... | Name of the element type(s) to be monitored. At least one element type must be present. All elements designations are generic and the "C", designation Connections, should be dropped. The exceptions for this are the CONROD and CONV element types.        |
| comp1 comp2...      | Name of the element component(s) to be monitored; e.g., etmax1 for max shear strain in the Z1 plane. The component names are defined in the Nastran Data Definition Language (NDDL). Also, the item codes from <a href="#">Item Codes, 1045</a> may be used. |
| ABSOLUTE(p)         | Print out the top p absolute values (Default for p is 5).  |
| MINALG(q)           | Print out the bottom q minimum algebraic values (Default for q is 5).  |
| MAXALG(r)           | Print out the top r maximum algebraic values (Default for r is 5).   |
| ALL(p)              | Print out all options: ABSOLUTE, MINALG, and MAXALG. (Default for p is 5).   |
| BRIEF               | Print out only the maxmin results (Default).   |
| FULL                | Print out the maxmin results followed by the standard data recovery format for the elements and grids at the retained set of the maximum or minimum occurrences.   |
| GLOBAL              | Selects the global coordinate system (see CD on the GRID entry) for monitoring grid point results.   |
| BASIC               | Selects the basic coordinate system for monitoring grid point results (Default).   |
| cid                 | Specifies a coordinate system ID for a system defined on a CORDij entry for monitoring grid point results.   |
| RMS                 | Print out the root-mean-square value of each maximum or minimum value requested by ABSOLUTE, MIN, or MAX over all time steps.  |
| CENTER              | Component selector when element allows for component name to occur in multiple places (Default).   |
| ALL                 | Selects all locations in an element where multiple locations exist.  |
| GROUP               | Reduces all occurrences of a component name to a single value before the action is performed.  |
| ENDS                | Selects the ends of a BEAM element ignoring intermediate stations.   |

**Remarks:**

1. MAXMIN(DEF) must be specified above all subcases, but this is not sufficient to request monitoring of maximums and minimums. The MAXMIN command must also be specified above or inside subcases.
2. MAXMIN(DEF) may be specified more than once.
3. Multiple element types may be grouped together, if the same component name is to be monitored across those types, by enclosing the element types in parentheses. This grouping does not combine the element types during processing. Each type and component action is performed in class.
4. Multiple component names may be compared collectively to the current maximum (or minimum), but only the maximum (or minimum) component in the group will be reported in the output. This is requested by enclosing the component names in parentheses.
5. Grid point component output is always converted to the basic coordinate system for monitoring when processing "sort1". The global system is the default when processing in "sort2".
6. Results for layers in composite elements, or intermediate stations in CBAR and CBEAM elements, are not supported.
7. Only real data recovery is supported.
8. When no MINA, MAXA, or ABSO keyword is supplied, the default values of p, q, and s will be 5. When any keyword is supplied, the other unreferenced keyword values will be set to zero, and no output will be created.
9. The component action keywords of CENTER, ALL, GROUP, and ENDS can only be applied to component names defined in the NDDL and occur at multiple places in element data recovery. They cannot be used with Item Codes.
10. Specify CQUAD4C and CTRIAC for corner stresses of CQUAD4 and CTRIA3 elements.
11. MAXMIN data output to the .op2 and .xdb files are not supported.
12. SORT1 and SORT2 output Case Control options (such as DISP(SORT2)=ALL), cause MAXMIN to behave differently.
  - SORT1 is only supported in SOLs 101, 103, 105, and 109. For each static loadcase, mode buckling eigenvalue, or time step, all the grids or elements will be searched for maximums and minimums.
  - SORT2 is only supported in SOL 112. For each selected grid or element, all the time steps will be searched for maximums and minimums.

**MCFRACTION**

## Modal Contribution Fractions Output Request

Requests the form and type of modal contribution fraction output.

Format:

$$\begin{aligned}
 & MCFRACTION([STRUCTURE], [PRINT, PUNCH][REAL or IMAG][PLOT], [(SORT = sorttype)], \\
 & [KEY = FRACTION_{sortitem}], [ITEMS = FRACTION_{ALL}_{(itemlist)}] [SOLUTION = ALL_m_{NONE}] [FILTER = 0.001_{fratio}] \\
 & [NULL = 12_{ipowr}]) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

Example:

```

SET 1001 = 10.0, 20.0, 100.0
SET 2001 = 716/T3, 809/T3, 412/T1
MCFRACTION (STRUCTURE,PRINT,PUNCH,ITEM=FRACTION,
             SORT=ABSD,KEY=PROJECTION,SOLUTION=1001)=2001
MCFRACTION (ITEMS=(FRACTION, PROJECTION), FILTER=0.01)=2001
  
```

| Descriptor | Meaning                                    |
|------------|--|
| STRUCTURE  | Request pertains to structure points only. |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output.

PHASE Requests polar format (magnitude and phase) of complex output.

| Descriptor | Meaning  |
|------------|--|
| SORT       | Keyword selecting one of the sort type options. Default is to produce output in increasing natural mode number order.  |
| sorttype   | One of the following modal contribution fraction output sorting options:<br>ABSA - output will be sorted by absolute value in ascending order.<br>ABSD - output will be sorted by absolute value in descending order.<br>ALGA - output will be sorted algebraically in ascending order.<br>ALGD - output will be sorted algebraically in descending order. |
| KEY        | Keyword selecting a sorting operation key quantity.  |
| sortitem   | Item from the item list table on which the SORT operation is performed. (Default is FRACTION.)   |
| ITEMS      | Keyword specifying data selection options.   |
| itemlist   | One (or more) of the following modal contribution fraction output items. If more than one item is selected, the list must be enclosed in parentheses:  |

| Item Identifier | Description   |
|-----------------|---|
| RESPONSE        | Each mode's response at each degree of freedom is selected.                     |
| PROJECTION      | Projection of modal response on solution.                                       |
| FRACTION        | Fraction of total displacement per mode (PROJECTION divided by total).          |
| SCALED          | Scaled magnitude (PROJECTION divided by largest magnitude of modal response).   |
| MODEDISP        | Modal displacements (complex solution at each DOF by mode number).              |
| MODERESP        | Modal response for each mode (polar format with respect to total displacement). |

|          |  |
|----------|--|
| SOLUTION | Keyword specifying the output solution time step, forcing frequency, or complex eigenvalue selections for which modal fractions will be generated (Default = ALL). |
| m        | Results for solutions in SET m will be output.   |
| FILTER   | Keyword specifying the value of the printed output data filter.  |
| fratio   | Value of output filter ratio (Default = 0.001).  |
| NULL     | Keyword specifying the power of ten used to detect a null response quantity.   |
| ipowr    | The power of ten used to detect a null response quantity (Default = 12).   |
| n        | Results for grid point components in SET n will be output.   |
| ALL      | Results for all solutions and/or grid point components will be output.   |
| NONE     | No modal contribution fractions will be output.  |

**Remarks:**

1. The MCFRACTION Case Control command is useful in modal frequency response (SOL 111), modal transient response (SOL 112), and modal complex eigenvalue analysis (SOL 110) solution sequences only. If superelements are used, its use is restricted to residual structure data recovery operations only. MCFRACTION is not supported in SOL 200 or 400.
2. Printed output includes results for all of the data items described in the itemlist table.
3. Punched output includes results for only the data items selected by the ITEMS keyword.
4. Modal contribution fractions are sorted by increasing order of mode number unless the SORT keywords specifies a particular sorting order. If a sorting order is specified, the KEY keyword selects the particular data item in the printed results tabular output listing that is sorted. When MODEDISP is selected, the magnitude is sorted. When MODERESP is selected, the real portion of the response is sorted.
5. The SOLUTION keyword can be used to select a subset of the solutions available. If SET m is specified, the items in the SET list are forcing frequency values, time step values, or complex eigenvalue mode numbers, depending upon the solution sequence used.
6. The FILTER keyword specifies a filter ratio value that is used to limit the amount of printed output produced. It applies to the data item selected by the KEY keyword if it is specified. If no KEY keyword is present, the default value of KEY=FRACTION will be used. The maximum value for the selected data item across all natural modes is determined. If the ratio of the data item value to the maximum data item value is less than fratio for any natural mode, no output for that natural mode is produced.
7. If the magnitude of the total response at a selected grid point component is less than  $1.0 \times 10^{-ipowr}$ , no modal contribution fraction output is generated for that degree of freedom. If ipowr is not in the range of 1 to 31, the default value of 12 is used for ipowr, producing a null response threshold of  $1.0 \times 10^{-12}$ .
8. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data has no meaning. Furthermore, projections of responses onto the total response are simply the individual modal contribution to the total response at a degree of freedom. Thus, the only items available for output are the individual modal response magnitude (PROJECTION), the modal fraction (FRACTION), and the scaled response magnitude (SCALED). Selection of any of the other items from the itemlist table causes selection of the modal response magnitude (PROJECTION) item.
9. A request of MCFRACTION output for a SET that contains no degrees-of-freedom in the analysis set, will result in the message: "SYSTEM WARNING MESSAGE 2001 (MCFRAC)"

## MCHSTAT

### Change State in SOL 600

Indicates which MCHSTAT Bulk Data entry option will be used to control the state variables (temperatures) used in this subcase for a thermal stress simulation in SOL 600.

Format:

MCHSTAT=N

Example:

MCHSTAT=10

| Descriptor | Meaning  |
|------------|--|
| N          | ID of a matching Bulk Data MSCHSTAT entry specifying temperatures or other state variable values to be used from a previous analysis for this subcase. |

Remark:

1. This entry may only be used in SOL 600.

**MEFFMASS**

## Modal Effective Mass Output Request

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.

Format:

$$\text{MEFFMASS} \left[ \begin{bmatrix} \text{PRINT} \\ \text{NOPRINT} \end{bmatrix} \begin{bmatrix} \text{PUNCH} \\ \text{NOPUNCH} \end{bmatrix}, \text{GRID} = \text{gid}, \begin{bmatrix} \text{SUMMARY}, \text{PARTFAC}, \\ \text{MEFFM}, \text{MEFFW}, \\ \text{FRACSUM}, \text{ALL} \end{bmatrix} \right] = \begin{cases} \text{YES} \\ \text{NO} \end{cases}$$

Examples:

MEFFMASS  
MEFFMASS (GRID=12, SUMMARY, PARTFAC)

| Descriptor | Meaning   |
|------------|---|
| PRINT      | Write output to the print file (Default).   |
| NOPRINT    | Do not write output to the print file.  |
| PUNCH      | Write output to the punch file.   |
| NOPUNCH    | Do not write output to the punch file (Default).  |
| gid        | Reference a grid point for the calculation of the rigid body mass matrix. The default is the origin of the basic coordinate system. |
| SUMMARY    | Requests calculation of the total effective mass fraction, modal effective mass matrix, and the rigid body mass matrix (Default).   |
| PARTFAC    | Requests calculation of modal participation factors.  |
| MEFFM      | Requests calculation of the modal effective mass in units of mass.  |
| MEFFW      | Requests calculation of the modal effective mass in units of weight.  |
| FRACSUM    | Requests calculation of the modal effective mass fraction.  |

Remarks:

1. The SUMMARY descriptor produces three outputs:

a. Modal effective mass matrix  $[\varepsilon^T][m][\varepsilon]$

where:

$\varepsilon$  = Modal participation factors:  $[m^1][\phi^T][M_{aa}][D_{ar}]$ .

$m$  = Generalized mass matrix.

$\phi$  = Eigenvectors.

$M_{aa}$  = Mass matrix reduced to the a-set (g-set for superelements).

$D_{ar}$  = Rigid body transformation matrix with respect to the a-set.

- b. A-set rigid body mass matrix:  $[D_{ar}^T][M_{aa}][D_{ar}]$ . For superelement this is computed at the g-set
  - c. Total effective mass Fraction: i.e., diagonal elements of the modal effective mass matrix divided by the rigid body mass matrix.
2. The PARTFAC describer outputs the modal participation factors table  $\varepsilon$ .
  3. The MEFFM describer outputs the modal effective mass table  $\varepsilon^2$ , the term-wise square of the modal participation factors table.
  4. The MEFFW describer outputs the modal effective weight table; i.e., the modal effective mass divided by PARAM, WTMASS.
  5. The FRACSUM describer outputs the modal effective mass fraction table; i.e., the generalized mass matrix (diagonal term) multiplied by the modal effective mass divided by the rigid body mass matrix (diagonal term).
  6. For superelements, the MEFFMASS Case Control command uses the residual structure eigenvalues and eigenvectors, by default. If, however, PARAM, FIXEDDB, -1 is specified, then the MEFFMASS Case Control command uses the component mode eigenvalues and eigenvectors.

**METHOD****Real Eigenvalue Extraction Method Selection**

Selects the real eigenvalue extraction parameters.

Format:

$$\text{METHOD} \begin{bmatrix} \text{BOTH} \\ \text{STRUCTURE} \\ \text{FLUID} \\ \text{COUPLED} \end{bmatrix} = n$$

Examples:

METHOD=33

METHOD(FLUID)=34

METHOD(COUPLED)=100

| Descriptor         | Meaning  |
|--------------------|--|
| BOTH               | The referenced EIGR or EIGRL Bulk Data entry will be applied to both the structure and the fluid portion of the model (Default).                           |
| STRUCTURE or FLUID | The referenced EIGR or EIGRL Bulk Data entry is applied to the structural or fluid portion of the model.   |
| COUPLED            | The referenced EIGR or EIGRL Bulk Data entry is applied to the structural and fluid coupled system of the model.   |
| n                  | Set identification number of an EIGR or EIGRL Bulk Data entry for normal modes or modal formulation, or an EIGB or EIGRL entry for buckling (Integer > 0). |

Remarks:

1. An eigenvalue extraction method must be selected when extracting real eigenvalues.
2. If the set identification number selected is present on both EIGRL and EIGR and/or EIGB entries, the EIGRL entry will be used.
3. METHOD(FLUID) and METHOD(STRUCTURE) permits a different request of EIGR or EIGRL for the fluid portion of the model in coupled fluid-structural analysis.
  - If METHOD(STRUCTURE) or METHOD(FLUID) is also specified, then they will override the METHOD(BOTH) selection.
  - The METHOD(FLUID) and METHOD(STRUCTURE) may be specified simultaneously in the same subcase for the residual structure only. Do not specify METHOD(FLUID) in a superelement subcase even if the superelement contains fluid elements.
  - The auto-omit feature (see [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide*) is not recommended. Therefore, only those methods of eigenvalue extraction that can process a singular mass matrix should be used; e.g., the EIGRL entry, or MGIV and MHOU on the EIGR entry.

4. METHOD(COUPLED) is only used when STRUCTURE and FLUID COUPLED problem. The corresponding EIGRL or EIGR requires either the frequency range of interest or ND (number of roots desired) entry. The real coupled modes computation is done with combination of Lanczos method and Subspace iteration method.

METHOD(COUPLED) is recommended for heavy fluid (like liquids) model. When it is applied to the light fluid model, it might require more coupled modes to get the correct result.

Real coupled method is not supported in SOL 200 and SOL 400. If Case Control METHOD(COUPLED) is used in SOL 200 or SOL 400, the following error message will be returned:

\*\*\* USER FATAL MESSAGE 22996 (SUBDMAP PREOPT)

CASE CONTROL COMMAND METHOD(COUPLED) IS NOT SUPPORTED IN SOL 200 and SOL 400.

**MFLUID**

## Fluid Boundary Element Selection

Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

**Format:**

MFLUID = n

**Example:**

MFLUID = 919

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of one or more MFLUID Bulk Data entries (Integer > 0). |

**Remark:**

1. For additional discussion, see [Coupled Fluid-Structure Analysis](#) in the *Dynamic Analysis User's Guide*.
2. Use parameter PARAM,VMOPT controls how the virtual mass is processed.

**MFREQUENCY****Master Frequency Set Selection Frequency Response**

Selects the set of forcing frequencies to be used in frequency analysis as master frequencies for K, B, and K4 matrix generation at each master frequency, in frequency dependent analysis.

Format:

$$MFREQUENCY\left(\begin{bmatrix} \text{Linear} \\ \text{Log10} \end{bmatrix}, [TOL= 0.1]\right) = \begin{pmatrix} \text{AUTO} \\ \text{NOAUTO} \\ n \end{pmatrix}$$

Example:

MFREQ (TOL=.15) ← Nastran will generate a master frequency list with TOL=.15

| Descriptor | Meaning   |
|------------|---|
| Linear     | Linear interpolation will be used between master frequencies to compute K, B, and K4 matrices found on selected Case Control FREQ entry. (Character, Default)   |
| Log10      | Log interpolation will be used between master frequencies to compute K, B, and K4 matrices found on selected FREQ entry. (Character)  |
| TOL        | For the default option AUTO, for frequency dependent TABLEDi entries associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries a master frequency will be selected whenever there is a TOL jump in any table value. A TOL=0.0 is the same as NOAUTO. (Real $\geq 0.0$ )                        |
| AUTO       | For frequency dependent analysis associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries, Nastran will automatically, depending on associated frequency dependent TABLEDi entries and the required Case Control FREQ=m entry, generate a series of master frequencies. (Character, Default). |
| NOAUTO     | If the user has supplied a Case Control MFREQ entry pointing to a FREQ, FREQ1, or FREQ2 bulk data entry, Nastran will use the user supplied entry and its associated frequency list.  |
| n          | Set identification number of FREQ, FREQ1, and FREQ2 Bulk Data entries. (Integer $> 0$ )   |

**Remark:**

1. Both direct frequency and modal frequency allow, through MAT1F, MAT2F, MAT8F, MAT9F, and PBUSHT, PELAST, PDAMPT entries frequency dependent analysis. The presence of any active MAT1F, MAT2F, MAT8F, or MAT9F entry or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries a master frequency will initiate the master frequency feature in frequency analysis.
2. MFREQ should be above or in the first subcase.
3. The K, B, and K4 matrices, at master frequencies, are formed in the standard way, using material and property values at the associated current master frequency.
4. For frequencies selected between master frequencies, interpolation of the K, B, and, K4 matrices are performed. These interpolated frequencies are selected using the required Case Control FREQ=m entry.
5. Interpolation between master frequencies  $K^{lm}$  and  $K^{2m}$  is  $K_{ij} = (d_2 K^{lm}{}_{ij} + d_1 K^{2m}{}_{ij})/(d_1 + d_2)$  where.

For Linear:

$$d1 = (f - f1); \quad d2 = (f2 - f)$$

For Log10:

$$d1 = (\log_{10}(f) - \log_{10}(f1)); \quad d2 = (\log_{10}(f2) - \log_{10}(f))$$

6. For master frequencies there is full displacement, velocity, acceleration type data recovery as well as full element force and stress type data recovery.
7. For interpolated frequencies, there is only (a-set) or modal (h-set) displacement, velocity, acceleration type data recovery. Because, the solution of interpolated frequencies is based on interpolated K, B, and K4, there is no element data recovery at interpolated frequencies.
8. For modal frequency analysis, associated with master frequencies, residual vector methods for viscous and structural damping have been greatly enhanced and orthogonalization techniques are employed which require the removal of free – free modes. For this reason the structure must be statically constrained with the use of SUPORT entries. Case Control selectable SUPORT1 entries may also be used.
9. For preloading (STATSUB), with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries, a fatal message will be issued.

\*\*\* USER FATAL MESSAGE 22711 (MTMD56)

FREQUENCY case control command missing for frequency response analysis.

STATSUB static loading is not currently supported for frequency dependent material/property.

10. PARAM, NMNLFRQ, REAL (Default=0.0) allows users to select material or property values for frequency dependent materials or properties at a desired frequency other than the “nominal” values specified on the MATi or PBUSH, PCOMP, PCOMPG, PDAMP, or PELAS entries. These values are used in determining the eigenvalue solution used in the modal frequency analysis.

11. If a user supplies a FREQ, FREQ1, or FREQ2 entry with a 0.0 value of forcing frequency the following message will be issued:

\*\*\* USER WARNING MESSAGE 22714 (SUBDMAP MFREQRS)

Forcing frequency value of 0.0 detected, nominal (initial) material properties used.

The rational for this selection is that 0.0 Hz is a static solution and SOL 101 would use material nominal.

12. Some examples of interaction between MFREQ=AUTO and an always required FREQ=m and interpolated values are as follows:

If frequency dependent TABLEDi entries associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries are detected, and the user has not specified any MFREQ Case Control entries, then Nastran will automatically scan the TABLEDi entries. Nastran does this by using the required FREQ=m entry and for any TABLEDi where there is a TOL jump between two consecutive FREQ=m frequencies determining a "Master frequency".

```

SOL111
CEND
FREQ = 10 ← Required
...
BEGIN BULK
FREQ1, 10, 10.0, 5.0, 78
...
MAT1, 2, 35.46,      , .49, 1.1-9
MAT1F, 2, 101,     103, 102 ← All 3 Tables for E, G, NU
required.
...
TABLED1, 101,
    ,.01, 7.3941, 50., 18.017, 100., 24.386, 200., 35.46,
    ,300., 44.45, 400., 51.264, 500., 56.321, ENDT
TABLED1, 102
    ,.01, 0.49, 1000., 0.49, ENDT
TABLED1, 103
    ,.01, 2.481, 50., 6.076, 100., 8.1832, 200., 11.899
    ,300., 14.916, 400., 17.203, 500., 18.90, ENDT
TABLED1, 200 ← Based on say an active PBUSHT entry.
    ,.01, 0.4, 50., 0.65, 100., 0.78, 200., 0.92
    ,300., 0.99, 400., 1.03, 500., 1.07, endt

```

#### **Internally Nastran Generates in Case Control:**

**MFREQ=11**

#### **Internally Nastran Generates in "Bulk Data":**

**FREQ, 11, 10., 15., 20., 30., 40., 50., 65.,
 ,85., 105., 130., 160., 190., 230., 275., 330.,
 ,400.**

Where for the above, Nastran has based on the user supplied FREQ=10 Case Control entry, scanned all the active user supplied TABLEDi entries, and whenever there is a jump in table value of 10% (default TOL), it creates an entry into the master frequency list MFREQ=11.

MFREQ will issue the message:

\*\*\*AUTOMATIC MASTER FREQUENCY LIST HAS BEEN SELECTED FOR  
SUBCASE ID 1:  
**FREQ**, 11., 10., 15., 20., 30., 40., 50., 65.,  
, 85., 105., 130., 160., 190., 230., 275., 330.,  
, 400.

**MODALKE****Modal Kinetic Energy Request**

Requests a modal kinetic energy calculation and specifies the output form.

**Format:**

$$\begin{aligned}
 & MODALKE \left[ \left( \begin{array}{l} SORT1 \\ SORT2 \end{array} \right) \left[ \begin{array}{l} PRINT \\ NOPRINT \end{array} \right] \left[ \begin{array}{l} PUNCH \\ PHASE \end{array} \right] \left[ \begin{array}{l} REAL \text{ or } IMAG \\ \end{array} \right] \left[ \begin{array}{l} ESORT = \left\{ \begin{array}{l} MODE \\ ASCEND \\ DESCENT \end{array} \right\} \end{array} \right] \right] \\
 & [THRESH = e] \left[ \left\{ \begin{array}{l} TIME \\ FREQ \end{array} \right\} = \left\{ \begin{array}{l} ALL \\ r \end{array} \right\} \right] \left[ \begin{array}{l} AVERAGE \\ AMPLITUDE \\ PEAK \end{array} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

**Examples:**

MODALKE= ALL  
SET 200= 1, 3, 4, 5, 7  
MODALKE (ESORT=DESCEND, THRESH=.001) = 200

| Descriptor   | Meaning   |
|--------------|---|
| SORT1        | Output will be presented as a tabular listing of modes for each frequency or time step.                               |
| SORT2        | Output will be presented as a tabular listing of frequencies or time steps for each mode.                             |
| PRINT        | Write the results to the .f06 file (Default).   |
| NOPRINT      | Do not write the results to the .f06 file.  |
| PUNCH        | Write the results to the punch (.f07) file.   |
| ESORT        | Present the modal energies sorted by mode number, ascending energy value or descending energy value (Default is MODE) |
| THRESH       | Write out only those energies greater than e (Default = 0.001).   |
| MODALKE      | Compute energies for all modes or the set of mode numbers defined in SET n (Default = ALL).                           |
| TIME or FREQ | Compute energies at all time steps, or frequencies, or the set of frequencies defined by SET r (Default = ALL).       |
| AVERAGE      | Requests average energy in frequency response analysis only (Default).  |
| AMPLITUDE    | Requests amplitude of energy in frequency response analysis only.   |
| PEAK         | Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.                  |
| ALL, n, NONE | Compute modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.                            |

**Remarks:**

1. Modal kinetic energy calculations will be limited to SOLs 112 (modal transient response) and 111 (modal frequency response).
2. The MODES descriptor selects from the set of the modes selected by the combination of the Case Control command MODESELECT, and user parameters PARAM,LMODES; PARAM,LFREQ; PARAM,HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) descriptor selects from the set of the time steps (or forcing frequencies) selected by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.
4. For frequency response analysis, the energy may be output as real or complex values. When REAL is specified, the output will be one of AVERAGE, AMPLITUDE or PEAK energy. (See Remark 4. of the EKE Case Control Command for a definition of these terms.) When IMAG or PHASE is specified, the output will be complex energy values.

**Output Format:**

The output formats for complex energy in frequency response analysis follow.

**For SORT1 option:**

| FREQUENCY = 2.100000E+01            |                     |                     |                     |
|-------------------------------------|---------------------|---------------------|---------------------|
| M O D A L K I N E T I C E N E R G Y |                     |                     |                     |
| (REAL/IMAGINARY)                    |                     |                     |                     |
| MODE NUMBER                         | ACTUAL              | NORMALIZED          | FRACTIONAL          |
| 1                                   | 1.171343E-04<br>0.0 | 1.409326E-02<br>0.0 | 1.065749E-02<br>0.0 |
| 2                                   | 2.049217E-05<br>0.0 | 2.465558E-03<br>0.0 | 1.864483E-03<br>0.0 |
| 3                                   | 5.175167E-04<br>0.0 | 6.226612E-02<br>0.0 | 4.708635E-02<br>0.0 |
| 4                                   | 1.732168E-05<br>0.0 | 2.084095E-03<br>0.0 | 1.576016E-03<br>0.0 |

This format repeats for each frequency.

**For SORT2 option:**

| MODE NUMBER = 1                     |                     |                     |                     |
|-------------------------------------|---------------------|---------------------|---------------------|
| M O D A L K I N E T I C E N E R G Y |                     |                     |                     |
| (MAGNITUDE/PHASE)                   |                     |                     |                     |
| FREQUENCY                           | ACTUAL              | NORMALIZED          | FRACTIONAL          |
| 1.000000E+00                        | 8.147644E-04<br>0.0 | 1.000000E+00<br>0.0 | 9.924866E-01<br>0.0 |
| 2.000000E+00                        | 4.066132E-03<br>0.0 | 1.000000E+00<br>0.0 | 9.936857E-01<br>0.0 |
| 3.000000E+00                        | 1.411670E-02<br>0.0 | 1.000000E+00<br>0.0 | 9.955440E-01<br>0.0 |

This format repeats for each mode number.

The output formats for AVERAGE energy in frequency response analysis follow. The particular type (AVERAGE, AMPLITUDE or PEAK) is identified in the output title.

For SORT1 option:

| FREQUENCY = 2.100000E+01                                    |              |              |              |  |
|---|--------------|--------------|--------------|--|
| M O D A L    K I N E T I C    E N E R G Y ( A V E R A G E ) |              |              |              |  |
| MODE NUMBER   | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 1   | 5.856715E-05 | 1.409326E-02 | 1.065749E-02 |  |
| 2   | 1.024608E-05 | 2.465558E-03 | 1.864483E-03 |  |
| 3   | 2.587583E-04 | 6.226612E-02 | 4.708635E-02 |  |
| 4   | 8.660840E-06 | 2.084095E-03 | 1.576016E-03 |  |

This format repeats for each frequency.

For SORT2 option:

| MODE NUMBER = 1   |              |              |              |  |
|---|--------------|--------------|--------------|--|
| M O D A L    K I N E T I C    E N E R G Y ( A V E R A G E ) |              |              |              |  |
| FREQUENCY   | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 1.000000E+00  | 4.073822E-04 | 1.000000E+00 | 9.924866E-01 |  |
| 2.000000E+00  | 2.033066E-03 | 1.000000E+00 | 9.936857E-01 |  |
| 3.000000E+00  | 7.058351E-03 | 1.000000E+00 | 9.955440E-01 |  |
| 4.000000E+00  | 2.872413E-02 | 1.000000E+00 | 9.977741E-01 |  |

This format repeats for each mode number.

The output formats for transient response analysis follow.

For SORT1 option:

| TIME STEP = 1.000000E+00                  |              |              |              |  |
|---|--------------|--------------|--------------|--|
| M O D A L    K I N E T I C    E N E R G Y |              |              |              |  |
| MODE NUMBER                               | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 1   | 1.483182E-02 | 3.610759E-01 | 2.316040E-01 |  |
| 2   | 4.107675E-02 | 1.000000E+00 | 6.414274E-01 |  |
| 3   | 2.165481E-04 | 5.271792E-03 | 3.381472E-03 |  |
| 4   | 1.387959E-05 | 3.378941E-04 | 2.167345E-04 |  |

This format repeats for each time step.

For SORT2 option:

| MODE NUMBER = 2                           |              |              |              |  |
|---|--------------|--------------|--------------|--|
| M O D A L    K I N E T I C    E N E R G Y |              |              |              |  |
| TIME STEP                                 | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 0.000000E+00                              | 0.0          | 0.0          | 0.0          |  |
| 9.999999E-02                              | 2.512383E-03 | 1.000000E+00 | 9.420497E-01 |  |
| 2.000000E-01                              | 8.475699E-03 | 1.000000E+00 | 9.566881E-01 |  |
| 3.000000E-01                              | 1.415435E-02 | 1.000000E+00 | 9.568216E-01 |  |

This format repeats for each mode number.

Processing:

Modal kinetic energy is calculated using the following relations:

For transient analysis,

$$[Actual\ KE] = \frac{1}{2} \{\dot{u}_h\}^T [M_{hh}] \{\dot{u}_h\}$$

For frequency response analysis,

$$[Actual\ KE] = \frac{1}{2} \{u_h\}^T [diag(\omega_i^2)] [M_{hh}] \{u_h\}$$

where  $\omega_i$  is the excitation frequency and output is complex. See Remark 4 of the [EKE \(Case\) Case Control](#) command for energy calculation relations when output is real.

[Normalized Kinetic Energy] = norm[Actual Kinetic Energy], normalized per column.

[Fractional Kinetic Energy] = [Normalized Kinetic Energy]/[diagonal [ $\{1.0\}^T$  [Normalized Kinetic Energy]]], term-by-term division.

**MODALSE**

## Modal Strain Energy Request

Requests modal strain energy calculation and specifies the output form.

Format:

$$\begin{aligned}
 & \text{MODALSE} \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right] \left[ \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right] \left[ \begin{array}{l} \text{PUNCH} \\ \text{REAL or IMAG} \end{array} \right] \left[ \begin{array}{l} \text{PHASE} \\ \text{ESORT} = \left\{ \begin{array}{l} \text{MODE} \\ \text{ASCEND} \\ \text{DESCENT} \end{array} \right\} \end{array} \right] \\
 & [\text{THRESH} = e] \left[ \begin{array}{l} \left\{ \begin{array}{l} \text{TIME} \\ \text{FREQ} \end{array} \right\} \\ r \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\} \left[ \begin{array}{l} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

Examples:

```
MODALSE= ALL
SET 100= 1, 3, 4, 5, 7
MODALSE (ESORT=ASCEND, THRESH=.0001) = 100
```

| Descriptor   | Meaning  |
|--------------|--|
| SORT1        | Output will be presented as a tabular listing of modes for each frequency or time step.                                |
| SORT2        | Output will be presented as a tabular listing of frequencies or time steps for each mode.                              |
| PRINT        | Write the results to the .f06 file (Default).  |
| NOPRINT      | Do not write the results to the .f06 file.   |
| PUNCH        | Write the results to the punch (.f07) file.  |
| ESORT        | Present the modal energies sorted by mode number, ascending energy value or descending energy value (Default is MODE). |
| THRESH       | Write out only those energies greater than e (Default = 0.001).  |
| MODALSE      | Compute energies for all modes or the set of mode numbers defined in SET n (Default = ALL).                            |
| TIME or FREQ | Compute energies at all time steps or frequencies or the set of frequencies defined by SET r (Default = ALL).          |
| AVERAGE      | Requests average energy in frequency response analysis only (Default).   |
| AMPLITUDE    | Requests amplitude of energy in frequency response analysis only.  |
| PEAK         | Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.                   |
| ALL, n, NONE | Compute modal energies for all modes, the modes defined on SET n, or (3) no modes.                                     |

**Remarks:**

1. Modal strain energy calculations will be limited to SOLs 112 (modal transient response) and 111 (modal frequency response).
2. The MODES descriptor selects from the set of the modes prescribed by the combination of Case Control command MODESELECT, and user parameters PARAM,LMODES; PARAM,LFREQ; and PARAM,HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) descriptor selects from the set of the time steps (or forcing frequencies) prescribed by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.
4. For frequency response analysis, the energy may be output as real or complex values. When REAL is specified, the output will be one of AVERAGE, AMPLITUDE or PEAK energy. (See Remark 4 of the ESE Case Control Command for a definition of these terms.) When IMAG or PHASE is specified, the output will be complex energy values.

**Output Format:**

The output formats for complex energy in frequency response analysis follow.

**For SORT1 option:**

| FREQUENCY = 2.100000E+01          |                     |                     |                     |
|-----------------------------------|---------------------|---------------------|---------------------|
| M O D A L S T R A I N E N E R G Y |                     |                     |                     |
| (REAL/IMAGINARY)                  |                     |                     |                     |
| MODE NUMBER                       | ACTUAL              | NORMALIZED          | FRACTIONAL          |
| 1                                 | 1.171343E-04<br>0.0 | 1.409326E-02<br>0.0 | 1.065749E-02<br>0.0 |
| 2                                 | 2.049217E-05<br>0.0 | 2.465558E-03<br>0.0 | 1.864483E-03<br>0.0 |
| 3                                 | 5.175167E-04<br>0.0 | 6.226612E-02<br>0.0 | 4.708635E-02<br>0.0 |
| 4                                 | 1.732168E-05<br>0.0 | 2.084095E-03<br>0.0 | 1.576016E-03<br>0.0 |

This format repeats for each frequency.

**For SORT2 option:**

| MODE NUMBER = 1                   |                     |                     |                     |
|-----------------------------------|---------------------|---------------------|---------------------|
| M O D A L S T R A I N E N E R G Y |                     |                     |                     |
| (MAGNITUDE/PHASE)                 |                     |                     |                     |
| FREQUENCY                         | ACTUAL              | NORMALIZED          | FRACTIONAL          |
| 1.000000E+00                      | 8.147644E-04<br>0.0 | 1.000000E+00<br>0.0 | 9.924866E-01<br>0.0 |
| 2.000000E+00                      | 4.066132E-03<br>0.0 | 1.000000E+00<br>0.0 | 9.936857E-01<br>0.0 |
| 3.000000E+00                      | 1.411670E-02<br>0.0 | 1.000000E+00<br>0.0 | 9.955440E-01<br>0.0 |

This format repeats for each mode number.

The output formats for AVERAGE energy in frequency response analysis follow. The particular type (AVERAGE, AMPLITUDE or PEAK) is identified in the output title.

For SORT1 option:

| FREQUENCY = 2.100000E+01                            |              |              |              |  |
|---|--------------|--------------|--------------|--|
| M O D A L S T R A I N E N E R G Y ( A V E R A G E ) |              |              |              |  |
| MODE NUMBER   | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 1   | 5.856715E-05 | 1.409326E-02 | 1.065749E-02 |  |
| 2   | 1.024608E-05 | 2.465558E-03 | 1.864483E-03 |  |
| 3   | 2.587583E-04 | 6.226612E-02 | 4.708635E-02 |  |
| 4   | 8.660840E-06 | 2.084095E-03 | 1.576016E-03 |  |

This format repeats for each frequency.

For SORT2 option:

| MODE NUMBER = | 1            | M O D A L S T R A I N E N E R G Y ( A V E R A G E ) |              |  |
|---------------|--------------|---|--------------|--|
| FREQUENCY     | ACTUAL       | NORMALIZED  | FRACTIONAL   |  |
| 1.000000E+00  | 4.073822E-04 | 1.000000E+00  | 9.924866E-01 |  |
| 2.000000E+00  | 2.033066E-03 | 1.000000E+00  | 9.936857E-01 |  |
| 3.000000E+00  | 7.058351E-03 | 1.000000E+00  | 9.955440E-01 |  |
| 4.000000E+00  | 2.872413E-02 | 1.000000E+00  | 9.977741E-01 |  |

This format repeats for each mode number.

The output formats for transient response analysis follow.

For SORT1 option:

| TIME STEP = 1.000000E+00          |              |              |              |  |
|-----------------------------------|--------------|--------------|--------------|--|
| M O D A L S T R A I N E N E R G Y |              |              |              |  |
| MODE NUMBER                       | ACTUAL       | NORMALIZED   | FRACTIONAL   |  |
| 1                                 | 1.483182E-02 | 3.610759E-01 | 2.316040E-01 |  |
| 2                                 | 4.107675E-02 | 1.000000E+00 | 6.414274E-01 |  |
| 3                                 | 2.165481E-04 | 5.271792E-03 | 3.381472E-03 |  |
| 4                                 | 1.387959E-05 | 3.378941E-04 | 2.167345E-04 |  |

This format repeats for each time step.

For SORT2 option:

| MODE NUMBER = | 2            | M O D A L S T R A I N E N E R G Y |              |  |
|---------------|--------------|-----------------------------------|--------------|--|
| TIME STEP     | ACTUAL       | NORMALIZED                        | FRACTIONAL   |  |
| 0.000000E+00  | 0.0          | 0.0                               | 0.0          |  |
| 9.999999E-02  | 2.512383E-03 | 1.000000E+00                      | 9.420497E-01 |  |
| 2.000000E-01  | 8.475699E-03 | 1.000000E+00                      | 9.566881E-01 |  |
| 3.000000E-01  | 1.415435E-02 | 1.000000E+00                      | 9.568216E-01 |  |

This format repeats for each mode number.

Processing:

Modal strain energy is calculated using the following relations:

$$[Actual\ SE] = \frac{1}{2} \{u_h\}^T [M_{hh}] \{u_h\}$$

For frequency response analysis real output, see Remark 4 of the [ESE \(Case\) Case Control command for energy calculation relations](#).

[Normalized Strain Energy] = norm[Actual Strain Energy], normalized per column.

[Fractional Strain Energy] = [Normalized Strain Energy]/[diagonal [ $\{1.0\}^T$  [Normalized Strain Energy]]], term-by-term division.

**MODES**

## Subcase Repeater

Repeats a subcase.

**Format:**

MODES=n

**Example:**

MODES=3

| Descriptor | Meaning  |
|------------|--|
| n          | Number of times the subcase is to be repeated (Integer > 0). |

**Remarks:**

1. This Case Control command can be illustrated by an example. Suppose stress output is desired for the first five modes only, and displacements for the next two modes, and forces for the remaining modes. The following example would accomplish this:

```
SUBCASE 1 $ FOR MODES 1 THRU 5
MODES = 5
STRESS = ALL
SUBCASE 6 $ FOR MODES 6 AND 7
DISPLACEMENTS = ALL
MODES = 2
SUBCASE 8 $ FOR MODE 8 AND REMAINING MODES
FORCE = ALL
```

2. This command causes the results for each mode to be considered as a separate, successively numbered subcase, beginning with the subcase number containing the MODES command. In the preceding example, this means that subcases 1, 2, 3, etc. are assigned to modes 1, 2, 3, etc., respectively.
3. If this command is not used, eigenvalue results are considered to be a part of a single subcase. Therefore, any output requests for the single subcase will apply for all eigenvalues.
4. All eigenvectors with mode numbers greater than the number of subcases defined in the Case Control Section are printed with the descriptors of the last subcase. For example, to suppress all printout for modes beyond the first three, the following Case Control commands could be used:

```
SUBCASE 1
MODES = 3
DISPLACEMENTS = ALL
SUBCASE 4
DISPLACEMENTS = NONE
BEGIN BULK
```

5. This command may be of no use in non-eigenvalue analysis and may cause data recovery processing to be repeated.

**MODESELECT****Computed Mode Selection**

Selects a subset of the computed modes for inclusion or exclusion in modal dynamic analysis.

**Format:**

Mode selection based on arbitrary mode numbers:

$$\text{MODESELECT}\left(\begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix}\right) = n$$

**Alternate Format 1:**

Mode selection based on number of lowest modes:

$$\text{MODESELECT}\left(\begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix}\right) \text{LMODES} = lm$$

**Alternate Format 2:**

Mode selection based on range of mode numbers:

$$\text{MODESELECT}\left(\begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix}\right) [\text{LMODENM} = lom][\text{HMODENM} = him]$$

**Alternate Format 3:**

Mode selection based on frequency range:

$$\text{MODESELECT}\left(\begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix}\right) [\text{LFREQ} = lof][\text{HFREQ} = hif][\text{UNCONSET} = m]$$

**Alternate Format 4:**

Mode selection based on modal effective mass fraction (MEFFMFRA) criteria:

$$\begin{aligned} \text{MODESELECT}\left(\begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \end{bmatrix}\right) & [\text{T1FR} [= t1fr]][\text{T2FR} [= t2fr]][\text{T3FR} [= t3fr]] \\ & [\text{R1FR} [= r1fr]][\text{R2FR} [= r2fr]][\text{R3FR} [= r3fr]] \\ & [\text{ALLFR} [= allfr]][\text{UNCONSET} = m] \begin{bmatrix} \text{SUM} \\ \text{ANYMIN} \\ \text{ALLMIN} \end{bmatrix} \end{aligned}$$

See Remark 14. for examples illustrating the use of the preceding formats.

| Descriptor         | Meaning  |
|--------------------|--|
| STRUCTURE          | References computed modes of the structure (Default).  |
| FLUID              | References computed modes of the fluid.  |
| $n > 0$            | Set identification of a previously appearing SET command. ONLY those modes whose mode numbers are in SET $n$ will be included in the analysis. If SET $n$ is not defined, then ONLY mode $n$ will be included in the analysis (Integer).   |
| $n < 0$            | $ n $ refers to the set identification of a previously appearing SET command. The modes whose mode numbers are in SET $ n $ will be EXCLUDED from the analysis. If SET $ n $ is not defined, then mode $ n $ will be EXCLUDED from the analysis (Integer).   |
| lm                 | Number of lowest modes that are to included. (Integer $> 0$ ).   |
| lom                | Lower limit of the mode number range for selecting the modes. See Remark 5. (Integer $> 0$ ).  |
| him                | Upper limit of the mode number range for selecting the modes. See Remark 5. (Integer $> lom > 0$ ).  |
| lof                | Lower limit of the frequency range for selecting the modes. See Remark 6. (Real $\geq 0.0$ ).  |
| hif                | Upper limit of the frequency range for selecting the modes. See Remark 6. (Real $> lof \geq 0.0$ ).  |
| UNCONSET           | Specifies a single mode or a set of modes for unconditional_inclusion or exclusion, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes.  |
| $m > 0$            | Set identification of a previously appearing SET command. Modes whose mode numbers are in SET $m$ will be included in the analysis, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes. If SET $m$ is not defined, then mode $m$ will be included in the analysis, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes (Integer).                             |
| $m < 0$            | $ m $ refers to the set identification of a previously appearing SET command. Modes whose mode numbers are in SET $ m $ will be excluded from the analysis, regardless of the selection criterion and regardless of the inclusion or exclusion of other modes. If SET $ m $ is not defined, then mode $ m $ will be excluded from the analysis, regardless of the selection criterion and regardless of the inclusion or exclusion of other modes (Integer). |
| TiFR / RiFR        | Flags explicitly listing components whose modal effective mass fraction (MEFFMFRA) values are to be considered for mode selection.   |
| <i>tifr / rifr</i> | Threshold values for the listed TiFR / RiFR components. See Remark 8. (0.0 $<$ Real $\leq 1.0$ ).  |
| ALLFR              | Flag indicating that the MEFFMFRA values of components not explicitly listed by the TiFR / RiFR flags are also to be considered for mode selection.  |

| Descriptor | Meaning   |
|------------|---|
| allfr      | Threshold value for components not explicitly listed by the TiFR / RiFR flags. See Remark 8. ( $0.0 < \text{Real} \leq 1.0$ ).  |
| SUM        | For each specified component, the modes are selected as follows:<br><br>The modes are first sorted in descending order of the corresponding MEFFMFRA values. Then, starting from the first mode in this sorted list, the modes are selected until the sum of corresponding MEFFMFRA values equals or just exceeds the threshold value for that component (Default). |
| ANYMIN     | Any mode whose MEFFMFRA value for any specified component equals or exceeds the threshold value for that component will be selected.  |
| ALLMIN     | Any mode whose MEFFMFRA values for all of the specified components equal or exceed the corresponding threshold values for those components will be selected.  |

**Remarks:**

1. This command is meaningful only in modal dynamic analysis (SOLs 110, 111, 112, 145, 146, and 200). It is ignored in all other analyses.
2. Only one MODESELECT command is allowed for the structure and only one MODESELECT command is allowed for the fluid and these should be specified above the subcase level.
3. The various formats of this command may not be combined.
4. The computed modes used for mode selection include the augmented modes (if any) resulting from residual vector calculations.
5. If LMODENM is specified without HMODENM, a default value of 10000000 (ten million) is assumed for HMODENM. If HMODENM is specified without LMODENM, a default value of 1 is assumed for LMODENM.
6. If LFREQ is specified without HFREQ, a default value of 1.0E+30 is assumed for HFREQ. If HFREQ is specified without LFREQ, a default value of 0.0 is assumed for LFREQ.
7. If the format involving the MEFFMFRA criteria is employed, it is not necessary to specify a MEFFMASS Case Control command or, even if such a command is specified, to explicitly request the calculation of the modal effective mass fractions. In the absence of such a command or request, the program will automatically perform the necessary calculations internally to ensure that the required modal effective mass fractions are computed.
8. If the T1FR / R1FR / ALLFR keywords are specified without the corresponding *tifr* / *rifr* / *allfr* threshold values, then a default value of 0.95 (that is, 95%) is assumed for these threshold values. If the selection criterion is SUM, and a default value of 0.05 (that is, 5%) is assumed if the selection criterion is ANYMIN or ALLMIN.
9. The modal effective mass for a given mode is a measure of how much mass is associated with that mode, and indicates the sensitivity of that mode to base excitation. Modal effective mass is meaningful only for fixed base modes. If a structure is not restrained, all the modal effective mass will be associated with its rigid body modes.

10. When the MODESELECT Case Control command is used in conjunction with the parameters LMODES/LMODESFL, LFREQ/LFREQFL, and HFREQ/HFREQFL, the hierarchy of their usage is as follows:
  - a. If there is a MODESELECT Case Control command, it takes precedence over the parameters LMODES/LMODESFL, LFREQ/LFREQFL, and HFREQ/HFREQFL. (It does not matter whether these parameters are defined directly via PARAM entries, or indirectly using the FLSFSEL Case Control command.)
  - b. If there is no MODESELECT Case Control command, then parameter LMODES/LMODESFL takes precedence over parameters LFREQ/LFREQFL and HFREQ/HFREQFL. In this case, the number of lowest modes specified by LMODES/LMODESFL will be included in the modal dynamic analysis.
  - c. If there is no MODESELECT Case Control command and no LMODES/LMODESFL parameter, then parameters LFREQ/LFREQFL and HFREQ/HFREQFL are honored. In this case, all of the computed modes whose frequencies are in the range specified by LFREQ/LFREQFL and HFREQ/HFREQFL will be included in the modal dynamic analysis.
  - d. If there is no MODESELECT Case Control command and no LMODES/LMODESFL, LFREQ/LFREQFL, or HFREQ/HFREQFL parameter, then all of the computed modes will be included in the modal dynamic analysis.
11. If a subset of the computed modes is selected for subsequent use in the modal dynamic analysis, the user is informed of this by a User Information Message. Also, a new eigenvalue table indicating the actual modes selected for the analysis is output. If the user has employed a MODESELECT command involving the MEFFMFRA criteria, the modal effective mass fractions for the selected modes are also output.
12. If the mode selection criterion results in no modes being selected for subsequent use in the modal dynamic analysis, the program terminates the job with a fatal message indicating that no modal formulation is possible.
13. If the use of the MODESELECT command results in the selection of all of the computed modes for subsequent use, the user is informed of this by a User Information Message.
14. The following examples illustrate the use of the various formats of the MODESELECT command described above.

#### Examples Illustrating Mode Selection Based on Arbitrary Mode Numbers:

```
$ INCLUDE ONLY STRUCTURE MODES 7, 9 AND 12 IN THE ANALYSIS
SET 100 = 7,9,12
MODESELECT = 100
$ EXCLUDE FLUID MODES 5 AND 6 FROM THE ANALYSIS
SET 200 = 5,6
MODESELECT (FLUID) = -200
$ EXCLUDE STRUCTURE MODE 5 FROM THE ANALYSIS
MODESELECT = -5 $ (SET 5 NOT DEFINED)
```

### Examples Illustrating Mode Selection Based on Number of Lowest Modes:

```
$ INCLUDE THE LOWEST 10 STRUCTURE MODES IN THE ANALYSIS
MODESELECT (LMODES = 10)
$ INCLUDE THE LOWEST 5 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID LMODES = 5)
```

### Examples Illustrating Mode Selection Based on Range of Mode Numbers:

```
$ INCLUDE ONLY STRUCTURE MODES 10 THRU 20 IN THE ANALYSIS
MODESELECT (LMODENM = 10 HMODENM = 20)
$ INCLUDE ALL STRUCTURE MODES HIGHER THAN THE 6th MODE
$ IN THE ANALYSIS
MODESELECT (LMODENM = 7)
$ INCLUDE THE LOWEST 10 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID HMODENM = 10)
```

### Examples Illustrating Mode Selection Based on Frequency Range:

```
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ IN THE RANGE OF 0.1 HZ. TO 100.0 HZ. IN THE ANALYSIS
MODESELECT (LFREQ = 0.1 HFREQ = 100.0)
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR BELOW 50.0 HZ., BUT INCLUDE THE 10th AND 11th
$ MODES REGARDLESS OF THEIR CYCLIC FREQUENCIES
SET 1000 = 10, 11
MODESELECT (HFREQ = 50.0 UNCONSET = 1000)
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR ABOVE 5.0 HZ., BUT EXCLUDE THE 6 MODE
$ REGARDLESS OF ITS CYCLIC FREQUENCY
MODESELECT (LFREQ = 5.0 UNCONSET = -6) $ SET 6 NOT DEFINED
```

### Examples Illustrating Mode Selection Based on Modal Effective Mass Fraction (MEFFMFRA) Criteria:

MODESELECT (T3FR)

The default selection criterion of SUM is assumed, and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T3.

As many modes with the highest MEFFMFRA(T3) values as possible, such that the sum of the values is equal to or just exceeds 0.95, will be selected.

MODESELECT (T1FR = 0.90 T2FR R3FR = 0.85)

The default selection criterion of SUM is assumed, and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T2.

As many modes with the highest MEFFMFRA(T1) values as possible, such that the sum of the values is equal to or just exceeds 0.90, will be selected.

Similarly, as many modes with the highest MEFFMFRA(T2) values as possible, such that the sum of those values is equal to or just exceeds 0.95, will be selected.

As many modes with the highest MEFFMFRA(R3) values as possible, such that the sum of those values is equal to or just exceeds 0.85, will be selected.

```
MODESELECT (T1FR T3FR = 0.10 UNCONSET = -6 ANYMIN)  
$ SET 6 NOT DEFINED
```

Since the selection criterion is specified as ANYMIN, a default value of 0.05 (5%) is assumed for the threshold value for component T1.

All modes, excluding mode 6, whose:

```
MEFFMFRA (T1) values are equal to or greater than 0.05 OR  
MEFFMFRA (T3) values are equal to or greater than 0.10
```

will be selected.

```
SET 1000 = 20, 30
```

```
MODESELECT (T2FR = 0.1 R3FR = 0.15 ALLFR UNCONSET = 1000 ALLMIN)
```

The ALLFR flag indicates that the T1, T3, R1, and R2 components which are not explicitly specified above must also be considered in mode selection. Since the selection criterion is specified as ALLMIN, a default value of 0.05 (5%) is assumed for the threshold value for these components.

All modes whose:

```
MEFFMFRA (T1) values equal or exceed 0.05 AND  
MEFFMFRA (T2) values equal or exceed 0.10 AND  
MEFFMFRA (T3) values equal or exceed 0.05 AND  
MEFFMFRA (R1) values equal or exceed 0.05 AND  
MEFFMFRA (R2) values equal or exceed 0.05 AND  
MEFFMFRA (R3) values equal or exceed 0.15 AND
```

will be selected.

Modes 20 and 30 will be selected regardless of their MEFFMFRA values.

**MODTRAK****Mode Tracking Request**

Selects mode tracking options in design optimization (SOL 200).

**Format:**

MODTRAK = n

**Example:**

MODTRAK=100

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification of a MODTRAK Bulk Data entry (Integer > 0). |

**Remark:**

1. Selection of a MODTRAK Bulk Data entry with the MODTRAK Case Control command activates mode tracking for the current subcase. This request is limited to normal modes subcases (ANALYSIS=MODES) in design optimization (SOL 200).

**MONCARL**

## Control for Monte-Carlo simulation

Control for Monte-Carlo simulation using the non-parametric variability method (NPVM). This Case Control command can be used in SOL 111 only.

**Format:**

```
MONCARL ( [NSAMPLE=m] , [SEED=k] , [OFFD= r] ) = n
```

**Example:**

```
MONCARL (NSAMPLE=100) =5
```

| Descriptor  | Meaning  |
|-------------|--|
| NSAMPLE = m | Specifies the number of random samples, m ( Integer >=0, Default = 0).                             |
| SEED = k    | Specifies the random number generator seed, k (Integer > 0, Default = 777).                        |
| OFFD = r    | Specifies the scale factor for off-diagonal entries of the random matrix, r (Real, Default = 1.0). |
| n           | Set identification number of a MONCARL Bulk Data entry (Integer > 0).                              |

**Remarks:**

1. Only one MONCARL command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. Keeping in mind the size and the naively parallel nature of Monte-Carlo problem typical simulation would be run in a parallel mode. The parallel DMP simulation is restricted to Multi-Master mode in order to get a good scalable performance. Multi-Master mode can be specified using "RUNOPT=MULTIMST", in "DOMAINSOLVER" Executive Control statement, e.g.:
 

```
DOMAINSOLVER ACMS (PARTOPT=DOF, RUNOPT=MULTIMST)
```
3. Running the Monte-Carlo simulation in Nastran would result in a concatenated OP2, HDF5 and PCH file containing the response results of each random sample of Monte-Carlo simulation. Each random sample would appear as a unique subcase in the OP2, HDF5 and PCH file.
4. PLT Viewer (provided with Nastran) can be used to read the result of Monte-Carlo simulation to generate meaningful response statistics. Currently the PLT Viewer can only read HDF5 and "PCH files in SORT 1 format" (not OP2).
5. HDF5 is the recommended output method for reading into PLT Viewer. Use NASTRAN system cell (702) HDF5=1 to request HDF5 output.

**MONITOR**

## Print Selection for Monitor Data

Specifies options in the printing of monitor data.

Format:

$$\text{MONITOR}[\text{REAL or IMAG}, \text{PHASE}, \text{NODSP1}, \text{NOPNT1}, \text{NOPNT2}, \text{NOPNT3}] = \begin{cases} \text{ALL} \\ \text{NONE} \end{cases}$$

Example:

```
MONITOR (PHASE, NOPNT1) =ALL
MONITOR (IMAG, NODSP1) =ALL
```

| Descriptor   | Meaning  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output (Default). |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                                       |
| NODSP1       | Do not include MONDSP1 results in the MONITOR point prints (default is to provide these prints).                                 |
| NOPNT1       | Do not include MONPNT1 results in the MONITOR point prints (default is to provide these prints).                                 |
| NOPNT2       | Do not include MONPNT2 results in the MONITOR point prints (default is to provide these prints).                                 |
| NOPNT3       | Do not include MONPNT3 results in the MONITOR point prints (default is to provide these prints).                                 |
| ALL          | Print all monitor point results, except for those deselected.  |
| NONE         | Do not print monitor point results   |

Remarks:

1. The MONITOR command is required in order to obtain MONITOR results in the printed output in SOLs 101, 103, 108, 109, 111, 112, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. SOL 144 does not require the MONITOR command and is ignored.
3. The MONITOR command should be above the subcase level or in the first subcase (above the step level for SOL 400). MONITOR commands in subcases subsequent to the first one or in any steps for SOL 400 are ignored.
4. MONPNT1 is not applicable in SOL 103.

5. The MONITOR command is not required for SOL 144 except in the special case of MONPNT3 with any exclusion flags other than SMAD. Monitor points with exclusion flags require special processing of the GRID point force data and are not recommended because of the potential excessive cost.
6. Results for dynamic solutions (SOLs 108, 109, 111, 112, 146 and 400 with NLTRAN) will be in SORT2 format. All other solutions will be in SORT1 format.

**Output Format:**

In frequency response analysis, there is no output format of complex energy.

**For SOLs 101, 103, and 144 - SORT1:**

```
S T R U C T U R A L   I N T E G R A T E D   F R E E   B O D Y   M O N I T O R   P O I N T   L O A D S (MOPNPT3)
MONITOR POINT NAME = FORCSM1                                     SUBCASE NO.      1
LABEL = SUMMATION OF FORCES AT GRID 9
CP   =          0           X =  8.00000E+01           Y =  0.00000E+00           Z =  0.00000E+00
AXIS   REST, APPLIED
-----
CX    1.600000E+01
CY    1.600000E+01
CZ    1.600000E+01
CMX   0.000000E+00
CMY   -3.200000E+02
CMZ   3.200000E+02
```

This format repeats for each monitor point and repeats for each subcase or mode.

**SOLs 108, 109, 111, 112 and 146 - SORT2:**

```
S T R U C T U R A L   I N T E G R A T E D   F R E E   B O D Y   M O N I T O R   P O I N T   L O A D S (MOPNPT3)
MONITOR POINT NAME = FORCSM1                                     COMPONENT = CX                                     SUBCASE NO.      1
LABEL = SUMMATION OF FORCES AT GRID 9
CP   =          0           X =  8.000000E+01           Y =  0.000000E+00           Z =  0.000000E+00
TIME STEP   RESULTANT
-----
0.000000E+00  0.000000E+00
1.000000E-02  7.692307E-01
2.000000E-02  1.301775E+00
3.000000E-02  1.137915E+00
4.000000E-02  6.557894E-01
5.000000E-02  9.679956E-01
6.000000E-02  1.354058E+00
7.000000E-02  9.230635E-01
8.000000E-02  6.696146E-01
9.000000E-02  1.178594E+00
9.999999E-02  1.275434E+00
```

This format repeats for each monitor point.

**MPC****Multipoint Constraint Set Selection**

Selects a multipoint constraint set.

**Format:**

MPC = n

**Example:**

MPC=17

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification number of a multipoint constraint set. This set identification number must appear on at least one MPC or MPCADD Bulk Data entry (Integer > 0). |

**Remarks:**

1. In cyclic symmetry analysis, this command must appear above the first subcase command.
2. Multiple boundary conditions (MPC sets) are not allowed in superelement analysis. If more than one MPC set is specified per superelement (including the residual), then the second and subsequent sets will be ignored.
3. In addition to select MPC/MPCADD bulk data entries, MPC=n can also be used to select a group of rigid elements for the analysis via SET3,n bulk data entry with RBEin or RBEEx in DES field of SET3 bulk data entry.
4. SET3,n is not needed if ALL rigid elements in the model are to be utilized.
5. Rigid element set section is supported in SOL 400 if 'RIGID=LINEAR' is present in case control deck.

**MPCFORCES****Multipoint Forces of Constraint Output Request**

Requests the form and type of multipoint force of constraint vector output.

**Format:**

$$\text{MPCFORCES} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{c} \text{PSDF}, \text{ATO}, \text{CRMS} \\ \text{or RALL} \end{array} \right], \\ \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{CID} \end{array} \right] \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

MPCFORCES=5

MPCFORCES (SORT2, PUNCH, PRINT, IMAG)=ALL

MPCFORCES (PHASE)=NONE

MPCFORCES (SORT2, PRINT, PSDF, CRMS, RPUNCH)=20

MPCFORCES (PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|               |  |
|---------------|--|
| REAL or IMAGE | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE         | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |

| Descriptor | Meaning   |
|------------|---|
| PSDF       | Requests that the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8. |
| ATOC       | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.             |
| CRMS       | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.                            |
| RALL       | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.  |
| RPRINT     | Writes random analysis results in the print file (Default).   |
| NORPRINT   | Disables the writing of random analysis results in the print file.  |
| RPUNCH     | Writes random analysis results in the punch file.   |
| CID        | Request the printing of output coordinate system ID in printed output file (.f06) file.   |
| ALL        | Multipoint forces of constraint for all points will be output. See Remarks 2 and 5.   |
| NONE       | Multipoint forces of constraint for no points will be output.   |
| n          | Set identification of a previously appearing SET command. Only multipoint constraint forces for points with identification numbers that appear on this SET command will be output (Integer > 0).  |

### Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
3. MPCFORCES=NONE overrides an overall output request.
4. In SORT1 format, MPCFORCEs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, the line will not be printed. If a sextet cannot be formed, then zero values may be output.
5. MPCFORCE results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).

6. MPCFORCE results due to linear elements only are available in SOL 129 with MDLPRM, MPCF129, 1.
7. In inertia relief analysis, the MPCFORCE output includes both the effects of applied and inertial loads.
8. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
9. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELO, ACCE, OLOAD, SPCForce and MPCF. In addition, the CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

**MPRES****Fluid Pressure Output Request**

Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.

**Format:**

$$MPRES \left[ \left( \begin{matrix} PRINT, PUNCH \\ PLOT \end{matrix} \right), \left( \begin{matrix} REAL \text{ or } IMAG \\ PHASE \end{matrix} \right) \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

MPRES=5

MPRES (IMAG) =ALL

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |   |
|--------------|---|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.                      |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| ALL          | Fluid pressures for all elements will be output.  |
| NONE         | Fluid pressures for no elements will be output.   |
| n            | Set identification number of a previously appearing SET command. Only fluid pressures for elements in this set will be output (Integer >0). |

**Remark:**

1. If PARAM,SPARSEDRL,NO is specified, then PARAM,DDRMM,-1 is also required in the modal solution sequences (SOLs 111, 112, 146, and 200).

**NLBUCK****Nonlinear Buckling Request**

Perform a nonlinear buckling analysis in SOL 400.

Format:

$$NLBUCK \left[ = \begin{cases} END \\ ALL \\ r \end{cases} \right]$$

Examples:

NLBUCK  
NLBUCK=END  
NLBUCK=ALL  
NLBUCK=0.3

| Descriptor | Meaning  |
|------------|--|
| END        | At the end of the step an eigenvalue projection is made to predict the buckled load (Default).   |
| ALL        | After each converged load increment within the step an eigenvalue projection is made to predict the buckled load.  |
| r          | After every multiple of r load step an eigenvalue projection is made to predict the buckled load. Buckling will also be computed at the final load increment. A tolerance of 1.E-6 is used to determine the load step's closeness to r. (Real) |

Remark:

1. The eigenvalue projection is only attempted after the first two converged load increments in order for there to be at least two tangent stiffness matrices for the projection.
2. NLBUCK may be specified in any STEP or SUBCASE.
3. In order to obtain the most accurate value for critical buckling factor, the user must set NO=1 or the smallest practical value on the NLSTEP entry or INTOUT=YES, ALL, or the largest practical value on the NLPARM entry. Specifying PARAM,LGDISP>0 is also recommended.
4. In addition to NLBUCK, a METHOD Case Control command may be specified to reference an EIGB, EIGR or EIGRL Bulk Data entry, or if asymmetric follower stiffness exists then a CMETHOD Case Control command may be specified to reference an EIGC entry. The EIGx entries should request the computation of at least the first mode. If no METHOD/CMETHOD command is present, a minimum of two eigenvalues are extracted for the buckling projection using an eigenvalue extraction method appropriate for the form of the tangent stiffness (symmetric or asymmetric). For manual control of follower stiffness symmetry see the description of user PARAMeters FOLLOWK and FKSYMFAC.

5. If NLBUCK=0.3 and the TOTTIME on the NLSTEP Buck Data entry is 1.0 then the buckling will be computed at load steps 0.3, 0.6, 0.9, and 1.0. Also, if the load factor is not a multiple of r then the next load factor will be used; for example, if TOTIME=1.0 and there are 50 load increments and r=.45 then buckling will be computed at load factors 0.46, 0.90, and 1.0.
6. There are three methods of eigenvalue extraction available for nonlinear buckling—Lanczos (EIGRL or EIGR entry with METHOD=LAN), enhanced inverse power method (EIGB entry with METHOD=SINV), and complex (EIGC entry) for unsymmetric stiffness due to follower stiffness.
  - a. If no METHOD or CMETHOD command is specified, then the program will automatically attempt to compute two modes (ND=2) with an unspecified values for the eigenvalue range (F1 and F2) using the Lanczos method.
  - b. The Lanczos method is recommended in most cases especially in finding the lowest mode.
  - c. If no modes can be found with no eigenvalue range was specified, then it is highly recommended that a range (L1 and L2 on EIGB, F1 and F2 on EIGR, and V1 and V2 on EIGRL) be specified.
  - d. If higher modes are desired, then the enhanced inverse power method is recommended with a narrow eigenvalue range specified for L1 and L2 on the EIGB entry.
  - e. If a METHOD command is specified but the stiffness is unsymmetric then User Warning Message 9430 will be issued.
7. Guidelines and limitations:
  - a. NLPARM Case Control command is not permitted in a nonlinear buckling step. NLSTEP must be specified in the nonlinear buckling step and KMETHOD=PFNT (default) is strongly recommended along with NO=1 for FIXED time stepping or INTOUT=YES for ADAPT time stepping. If KMETHOD=ITER then KSTEP=1 is strongly recommended.
  - b. It is strongly recommended that PARAM,LGDISP,1 is specified.
  - c. RIGID=LAGRANGE or LGELIM with advanced nonlinear elements is not supported in nonlinear buckling analysis.
  - d. It is strongly recommended to specify an eigenvalue range on the EIGR, EIGRL, EIGB, and EIGC Bulk Data entries. But with NLBUCK=ALL it may be difficult to define a range for all load increments.
  - e. Node-to-segment is not recommended except in the case of permanent glue contact.

## NLHARM

## Nonlinear Harmonic Analysis Parameter Selection

Selects the parameters used for nonlinear harmonic response analysis.

**Format:**

NLHARM = n

**Example:**

NLHARM = 79

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of NLHARM Bulk Data entry. (Integer > 0) |

**Remarks:**

1. This entry references the NLHARM Bulk Data entry which defines the number of subharmonics and harmonics to be used in the analysis as well as referencing the excitation frequency specification.
2. Gyroscopic terms are included if the RGYRO Case Control command and Bulk Data entry are specified. If gyroscopic terms are desired, the user must specify the constant rotor speed (ASYNC option) for the reference rotor.

**NLIC**

## Nonlinear Initial Condition or Preload

Selects a previously executed load increment as the initial conditions or preload for a nonlinear or perturbation STEP in SOL 400.

Format:

$$\text{NLIC} \left[ \text{SUBCASE } i \left[ , \text{STEP } j \left[ , \begin{array}{c} \text{LOADFACT} \\ \text{TIME} \end{array} \right] \left[ \begin{array}{c} (T = \text{tol}) \\ \text{NEAR} \end{array} \right] f \right] \right]$$

Example:

```
NLIC SUBCASE 100 STEP 10 LOADFACT 0.8
NLIC SUBCASE 100 STEP 10 LOADFACT (t=0.01) 0.66
NLIC STEP 10 TIME 12.0
NLIC TIME NEAR 7.8
```

| Descriptor | Meaning  |
|------------|--|
| SUBCASE    | Keyword to select the SUBCASE ID   |
| i          | Specifies the identification number of a previously executed subcase. (Integer > 0; Default is the subcase where the current NLIC is located).   |
| STEP       | Keyword to select the STEP ID.   |
| j          | Specifies the identification number of a previously executed STEP (Integer > 0, Default is the last STEP).                                       |
| LOADFACT   | Keyword to select load factor or time. Both keywords are equivalent.   |
| TIME       |  |
| f          | Specifies the load factor of a previously executed load increment in linear or nonlinear static analysis (Real > 0.0, see Remark 3 for default). |
| T          | Keyword to select the tolerance.   |
| NEAR       | Keyword for the nearest load factor or time to f.  |
| tol        | Specified the load factor or time tolerance to f. (Real $\geq 0.0$ ; Default = 1.0E-06)  |

Remarks:

1. The NLIC command can only point to a load increment whose output flag is on - an available restart point in the static analysis. If NLIC is not pointing to an available restart point, a fatal error will be issued and the job will be terminated.
2. The NLIC can be used in nonlinear static, nonlinear transient and perturbation analysis. If SUBCASE referred in NLIC is not the current SUBCASE, this usage is not recommended. The system(779) = 1 may be used to avoid this usage, which stops the job in this case.

3. For nonlinear transient analysis (ANALYSIS=NLTRAN), NLIC can only appear in the first transient analysis STEP in a SUBCASE. (Note that the first transient analysis step may not be the first step of subcase.) Otherwise, it will be ignored and a warning message will be issued.
4. For linear perturbation analysis, NLIC must point to a previous NLSTATIC step in the same SUBCASE.
5. If NLIC is specified without any of the keywords, or NLIC is not present in a nonlinear STEP or perturbation STEP, the initial condition or PRELOAD is taken from the last available restart point in the immediate previous static step.
6. In the same nonlinear transient step, NLIC cannot appear together with the IC Case Control command. A fatal error message will be issued if NLIC and IC appear in the same step. Please note that IC is meaningful only in the first STEP of a SUBCASE, and the step is a nonlinear transient analysis.
7. When Keyword NEAR is used, the search for the nearest load factor or time is limited inside the specified SUBCASE i STEP j.
8. NLIC can be used in SOL 400 only.
9. If NLIC is used with ANALYSIS=NLSTATIC it may cause convergence problems. If NLIC is used across different subcases: the initial conditions, the deformation, the strains, the stresses, etc., are carried over between Subcases. However, the load is NOT carried over. This can cause an unbalanced load condition that may cause convergence problems.

**NLLOAD****Nonlinear Load Output Request**

Requests the form and type of nonlinear load output for transient problems.

Format:

$$NLLOAD[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Example:

NLLOAD=ALL

| Descriptor | Meaning   |
|------------|---|
| PRINT      | The printer will be the output medium.  |
| PUNCH      | The punch file will be the output medium.   |
| ALL        | Nonlinear loads for all solution points will be output.   |
| NONE       | Nonlinear loads will not be output.   |
| n          | Set identification of a previously appearing SET command. Only nonlinear loads for points with identification numbers that appear on this SET command will be output (Integer > 0). |

Remarks:

1. Nonlinear loads are output only in the solution (d or h) set.
2. The output is available in SORT2 format only.
3. Both PRINT and PUNCH may be used.
4. NLLOAD=NONE allows overriding an overall output request.

**NLOPRM**

## Nonlinear Regular and Debug Output Control Parameters - SOL 400

Controls MSC Nastran nonlinear solution output, debug printout, debug POST and punch-out of contact constraints of MPC and MPCY Bulk Data entries.

**Format:**

*NLOPRM* = [*OUTCTRL* = {*STD,SOLUTION,INTERM*}]

$$\begin{aligned} NLDBG &= \left[ \begin{array}{l} NONE \\ NLBASIC,NRDBG,ADVDBG, \left\{ \begin{array}{l} N3DBAS \\ N3DMED \\ N3DADV \\ N3DSUM \end{array} \right\} \end{array} \right] \\ DBGPOST &= \left[ \begin{array}{l} NONE \\ LTIME \\ LSTEP \\ LSUBC \\ ALL \end{array} \right], MPCPCH = \left[ \begin{array}{l} NONE \\ BEGN,OTIME,STEP \\ YBEGN,YOTIME,YSTEP \end{array} \right] \\ DELIMIT &= \left\{ \begin{array}{l} No \\ Yes \end{array} \right\} \\ GRIDINF &= \left\{ \begin{array}{l} No \\ MAXGRID \\ GID \end{array} \right\} \end{aligned}$$

**Example(s):**

NLOPRM            OUTCTRL=STD, SOLUTION            DBGPOST=LTIME  
 NLOPRM            OUTCTRL=(SOLUTION, INTERM), MPCPCH=(OTIME, STEP)

| Descriptor | Meaning  |
|------------|--|
| OUTCTRL    | Selects one or more following nonlinear solution output options  |
| STD        | Standard Nastran output (Default)  |
| SOLUTION   | Solution set output which does not include solutions for superelements   |
| INTERM     | Intermediate nonlinear solution output in OP2 and HDF5 files, at each solution output interval, set by the user through nonlinear solution controls, such as NLPARM. |
| NLDBG      | Keyword selecting one or more debug printout options   |
| NONE       | No debug print output (Default)  |
| NLBASIC    | Basic nonlinear information printout   |
| NRDBG      | Newton-Raphson iteration information printout  |

| Descriptor | Meaning  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
|------------|--|------|---------------------------------------|-------|---|-------|---|-------|---|-------|---|--------|--|-------|---|
| ADVDBG     | Advance nonlinear debug printout   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| N3DBASE    | For 3D contact, it requests to print out the error tolerance of each contact body in the F06 file.   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| N3DMED     | For 3D contact, it requests to print out the summary table of all contact parameters, in addition to the printout by N3DBAS, in the .f06 file.   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| N3DADV     | For 3D contact, it requests to print out the body contact information in long form, in addition to the printouts by both N3DBAS and N3DMED, in the F06 file.   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| N3DSUM     | For 3D Contact, it simplifies body contact information in N3DADV.  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| DBGPOST    | Selects one of the following debug POST options <table> <tr> <td>NONE</td><td>No output (Default)</td></tr> <tr> <td>LTIME</td><td>To output all iterations in the last load or time increment</td></tr> <tr> <td>LSTEP</td><td>To output all iterations in the last STEP</td></tr> <tr> <td>LSUBC</td><td>To output all iterations in the last SUBCASE</td></tr> <tr> <td>ALL</td><td>To output all iterations</td></tr> </table>   | NONE | No output (Default)                   | LTIME | To output all iterations in the last load or time increment | LSTEP | To output all iterations in the last STEP     | LSUBC | To output all iterations in the last SUBCASE                              | ALL   | To output all iterations                                |        |  |       |   |
| NONE       | No output (Default)  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| LTIME      | To output all iterations in the last load or time increment  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| LSTEP      | To output all iterations in the last STEP  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| LSUBC      | To output all iterations in the last SUBCASE   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| ALL        | To output all iterations   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| MPCPCH     | Selects one or more following options to punch out contact constraint equations in terms of MPC or MPCY Bulk Data entries in Nastran PCH file. This is not available for segment-to-segment contact. <table> <tr> <td>NONE</td><td>No MPC or MPCY punch output (Default)</td></tr> <tr> <td>BEGN</td><td>MPC punch in the beginning of the very first iteration</td></tr> <tr> <td>OTIME</td><td>MPC punch at every user requested output step</td></tr> <tr> <td>STEP</td><td>MPC punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE</td></tr> <tr> <td>YBEGN</td><td>MPCY punch in the beginning of the very first iteration</td></tr> <tr> <td>YOTIME</td><td>MPCY punch at every user requested output step</td></tr> <tr> <td>YSTEP</td><td>MPCY punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE)</td></tr> </table> | NONE | No MPC or MPCY punch output (Default) | BEGN  | MPC punch in the beginning of the very first iteration      | OTIME | MPC punch at every user requested output step | STEP  | MPC punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE | YBEGN | MPCY punch in the beginning of the very first iteration | YOTIME | MPCY punch at every user requested output step | YSTEP | MPCY punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE) |
| NONE       | No MPC or MPCY punch output (Default)  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| BEGN       | MPC punch in the beginning of the very first iteration   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| OTIME      | MPC punch at every user requested output step  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| STEP       | MPC punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| YBEGN      | MPCY punch in the beginning of the very first iteration  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| YOTIME     | MPCY punch at every user requested output step   |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| YSTEP      | MPCY punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE)  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| DELIMIT    | Select output of delimiter for subcases, steps, increments, and iterations. <table> <tr> <td>No</td><td>No output (default)</td></tr> <tr> <td>Yes</td><td>Select output</td></tr> </table>  | No   | No output (default)                   | Yes   | Select output   |       |   |       |   |       |   |        |  |       |   |
| No         | No output (default)  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| Yes        | Select output  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |
| GRIDINF    | Select output of grid point information (1) displacements of the grid; (2) EID: element ID connected to this grid; (3) PID: property ID of relevant elements; (4) ID of contact body which the grid belongs to; and, (5) if the grid locates on the surface of contact body. <table> <tr> <td>No</td><td>No output (default)</td></tr> </table>  | No   | No output (default)                   |       |   |       |   |       |   |       |   |        |  |       |   |
| No         | No output (default)  |      |                                       |       |   |       |   |       |   |       |   |        |  |       |   |

| Descriptor | Meaning   |
|------------|---|
| MAXGRID    | Output information of grid which has maximum displacement component |
| GID        | Output information of grid which user specifies                     |

**Remarks:**

1. This Case Control command is used to better control nonlinear solution output during solution process, to provide MSC users a direct access to nonlinear solutions even the job is still running, to give the users some tools to debug the nonlinear solution process and gain some insight of nonlinear solution procedure, and to allow users to print out MPC and MPCY equations from contact constraints before and during solution process.
2. OUTCTRL=SOLUTION is designed to debug nonlinear analysis which may fail during the analysis. For other selections, some results such as displacement, SPC force, and so on, are written into F06 file only after the analysis is finished completely. This is helpful for user to understand the nonlinear analysis when job is terminated abnormally. With OUTCTRL=SOLUTION, only part of the results will be output. Availability of output request for debugging purposes are summarized at the following table.

| Output Request | OP2 | F06 |
|----------------|-----|-----|
| BOUTPUT        | NO  | YES |
| DISPLACEMENT   | YES | YES |
| GPFORCE        | NO  | NO  |
| MPCFORCES      | YES | YES |
| NLSTRESS       | NO  | YES |
| OLOAD          | YES | YES |
| SPCFORCES      | YES | YES |
| STRESS         | NO  | NO  |
| STRAIN         | NO  | NO  |

Please note that OP2 file only includes the results at the end time of every load step, instead of the user-specified output intervals assigned in NLSTEP or NLPARM. It is recommended to use OUTCTRL=SOLUTION,INTERM to get nonlinear solutions in both F06 and intermediate OP2 files for debugging purposes.

3. For OUTCTRL = INTERM, the intermediate output is only available in OP2 file. GPFORCE is not available in intermediate OP2 files and can be obtained by OUTCTRL=STD.
4. For NLDBG, all nonlinear information is printed out in only F06 file.
5. For DBGPOST, all nonlinear information is printed out in the MASTER/DBALL.
6. For MPCPCH, it is a punch output that MPC and MPCY requests cannot be mixed.

7. In MPCPCH punch output, it is restricted that SID=1 for a general contact and SID=2 for the glued contact.
8. When using MPCPCH for the permanent glued contact, user may specify any keyword of BEGN, STEP and OTIME to punch output MPC type of constraints, or any keyword of YBEGN, YSTEP and YOTIME to punch out MPCY type of constraints.
9. The output to the file is also influenced by the NLPACK Param. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2  
, fixed, 2000, 20  
, mech, u
```

It has total 2000 increments, and is asked output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With "intermediate output request, you will have only one OP2 file.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, you will have 100 OP2 files.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, you will have 50 OP2 files.

## NLPARM

## Nonlinear Static Analysis Parameter Selection

Selects the parameters used for nonlinear static analysis.

**Format:**

NLPARM = n

**Example:**

NLPARM=10

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of NLPARM and NLPCI Bulk Data entries (Integer > 0). |

**Remarks:**

1. NLPARM and NLPCI Bulk Data entries will not be used unless selected.
2. NLPARM may appear above or within a subcase.
3. For SOL 600, the only fields used are ID, NINC, DT (creep only), EPSU, EPSP, and EPSW. Use PARAM,MARCOTIM instead of INTOUT. For other fields, advanced convergence controls are available using the NLAUTO, NLSTRAT, and PARAM,MARCDEF Bulk Data entries.

**NLRESTART**

## Nonlinear Restart Request

Request a RESTART execution at a specified point for SOL 400.

Please note that when running NLRESTART with Advanced Nonlinear Element, the Data Blocks for Advanced Nonlinear Element must be saved in the corresponding cold start run with the proper DBSAVE, NLPACK, and INTOUT. See Remark 12.

**Format:**

$$\text{NLRESTART} \left[ \text{SUBCASE } i \left[ , \text{STEP } j \left[ , \begin{array}{c} \text{LOADFAC } f \\ \text{TIME } t \end{array} \right] \right] \right] [\text{OVERRIDE}]$$

**Example:**

```
NLRESTART SUBCASE 1, STEP 2, LOADFAC 0.3
```

| Descriptor | Meaning  |
|------------|--|
| i          | Specifies the identification number of a previously executed SUBCASE (Integer; Default is the first SUBCASE).  |
| j          | Specifies the identification number of a previously executed STEP (Integer; Default is the first STEP).  |
| f          | Specified the load factor of a previously executed load increment in nonlinear static analysis (Real; $0.0 \leq f \leq 1.0$ ; Default = 1.0).  |
| t          | Specified the time of a previously executed time step in nonlinear transient analysis (Real; $t_0 \leq t \leq t_n$ , where $t_0$ is the initial time of STEP j, and $t_n$ is the last time of STEP j; Default = $t_n$ ).   |
| OVERRIDE   | To force the run to continue even though there are fundamental changes to the model's geometry, properties, and/or connectivity. In general, NLRESTART does not allow changes to the model's geometry, properties, or connectivity and UFM 9424 will be issued if a change is detected. Please be cautious in using OVERRIDE because the model changes may result in wrong answers and/or a fatal termination and, therefore, is not recommended in all cases. |

**Remarks:**

1. The NLRESTART command can be used in SOL 400 (NONLIN) only.
2. The NLRESTART command must appear before any SUBCASE command.
3. To perform a restart, the data base for the original run must be made available by using the ASSIGN File Management statement or other equivalent method.

4. The restart run can only be executed at a load increment (or time step) whose output flag is on an available restart point. (See the field INTOUT on the NLPARM Bulk Data entry, and NO on ). When a user-specified restart point is not available, the closest previous restart point that is available will be applied automatically.
5. If only NLRESTART is specified, a restart begins from the last available restart point in the previous run. Otherwise, at least one set of the SUBCASE i, STEP j, or LOADFC f (or TIME t) must be specified.
6. In static analysis, f is reset to 0.0 when  $f < 0.0$ , and it makes the restart begin from the beginning of STEP j. f is reset to 1.0 when  $f > 1.0$ , which makes the restart begin from the beginning of the next STEP (after STEP j).
7. In transient analysis, t is reset to  $t_0$  when  $t < t_0$ , and it makes the restart begin from the beginning of STEP j. t is reset to  $t_n$  when  $t > t_n$ , which makes the restart begin from the beginning of the next STEP (after STEP j).
8. The NLRESTART Case Control command must contain all of the commands used in the original execution up to the point where the restart is requested.
9. All data contained on the database from the restart point will be deleted when the restart begins.
10. When using NLRESTART, its corresponding COLD START input model should not use "PARAM,DBALL,SCRATCH", which will remove the database required for NLRESTART after completion of COLD START run.
11. NLRESTART is used for Nonlinear Restart Request. The restart step must be a nonlinear analysis, NLSTAT or NLTRAN. NLRESTART from perturbation step or thermal analysis step is not supported in SOL400.
12. DBSAVE in the Case Control Section is used to control saving of datablocks of advanced nonlinear elements for static and transient nonlinear analysis in SOL 400.
  - 1 No datablocks of advanced nonlinear elements is saved
  - 0 Saving datablocks of advanced nonlinear elements at the end of each loadcase (default)
  - >0 Saving datablocks of advanced nonlinear elements at the every nth output request of results

In DBSAVE=0, NLRESTART may start at the end of the loadcase (load step) for advanced nonlinear elements. If DBSAVE=n (>0), NLRESTART may start at the every INTOUT\* n th output point.

For NLTRAN analysis in SOL 400, the output to the file, i.e., NLRESTART usage, is also influenced by the NLPACK Param. For example, one NLTRAN analysis has NLSTEP as:

NLSTEP, 900, 0.2

,fixed, 2000, 20

,mech, u

It has total 2000 increments, and is asked output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, you will have only one OP2 file.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With intermediate output request, you will have 100 OP2 files.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With intermediate output request, you will have 50 OP2 files.

Similarly, NLRESTART can be started at those output points ONLY.

**NLSTEP**

## Nonlinear Control Parameters for Mechanical, Thermal, and Coupled Analysis Selection

Selects integration and output time steps for static and transient nonlinear analysis in SOL 400.

**Format:**

NLSTEP=n

**Example:**

```
NLSTEP=10
SUBSTEP=1
ANALYSIS=HSTAT
SUBSTEP=2
ANALYSIS=NLSSTAT
```

| Descriptor | Meaning   |
|------------|---|
| n          | Identification number of a NLSTEP Bulk Data entry. (Integer > 0). |

**Remarks:**

1. An NLSTEP entry can be selected to execute a nonlinear static or nonlinear transient analysis in SOL 400. A NLSTEP is used in lieu of a NLPARM or both.
2. If a NLSTEP is present *anywhere* in a SUBCASE, then any NLPARM or entries in the SUBCASE *will be ignored*.
3. When used for coupled analysis, the NLSTEP must be above the first SUBSTEP command. A *single* NLSTEP entry is *used for all SUBSTEPs* of the STEP. (See the above example for use with coupled analysis.)

**NLSTRESS**

## Nonlinear Element Stress Output Request

Requests the form and type of nonlinear element stress output in SOLs 106 and 400.

Format:

$$NLSTRESS \left[ \begin{pmatrix} SORT1 \\ SORT2 \end{pmatrix}, \begin{pmatrix} PRINT & PUNCH \\ PLOT \end{pmatrix}, [NLOUT = m] \right] = \left\{ \begin{array}{c} ALL \\ n \\ See \text{ Remark } 3 \end{array} \right\}$$

Examples:

NLSTRESS=5  
 NLSTRESS (SORT1,PRINT,PUNCH,PHASE)=15  
 NLSTRESS (PLOT)=ALL  
 NLSTRESS (NLOUT=23)=ALL

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of elements for each load.     |
| SORT2      | Output will be presented as a tabular listing of load for each element type. |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|     |  |
|-----|--|
| ALL | Stresses for all nonlinear elements will be output.  |
| n   | Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output (Integer > 0). |

| Descriptor | Meaning   |
|------------|---|
| NONE       | No nonlinear element stress will be output (Default).                           |
| NLOUT      | For SOL 400 only. Allows the selection of additional types of nonlinear output. |
| m          | Identification of a NLOUT Bulk Data entry. (Integer > 0)                        |

**Remarks:**

1. ALL should not be used in a transient problem due to potentially excessive output.
2. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.
3. If there is a stress request, the default output set is that of the STRESS request.
4. For NLTRAN analysis, NLSTRESS does not support SORT1 selection.

## NONLINEAR

### Nonlinear Dynamic Load Set Selection

Selects nonlinear dynamic load set for transient response or nonlinear harmonic response problems.

**Format:**

NONLINEAR = n

**Example:**

NONLINEAR=75

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of NOLINI, NLRGAP or NLRSFD Bulk Data entry (Integer > 0). |

**Remark:**

1. Nonlinear force Bulk Data entries (NOLINI, NLRGAP or NLRSFD) will be ignored unless selected in the Case Control Section.
2. At least one degree of freedom must be defined on a nonlinear force entry and called up by the NONLINEAR Case Control command in nonlinear harmonic response.

## NOUTPUT

## Normal Output Request in Cyclic Symmetry Problems

Requests physical output in cyclic symmetry problems.

Format:

$$NOUTPUT \left\{ k, \begin{array}{c} R \\ L \end{array} \right\} = \left\{ \begin{array}{c} ALL \\ m \end{array} \right\}$$

Examples:

NOUTPUT (R)=ALL

NOUTPUT (2)=5

NOUTPUT (4,L)=10

| Descriptor | Meaning  |
|------------|--|
| ALL        | Output for all segments is desired.  |
| m          | Output for segments specified in SET m is desired (Integer > 0).   |
| k          | Used in eigenvalue analysis to request eigenvector and internal force output for harmonics specified in SET k (Integer > 0).     |
| R, L       | Output for only the right- or left-half of segments specified as ALL or in SET m.<br>R and L are used in dihedral symmetry only. |

Remarks:

1. Sets k and m are defined on SET commands.
2. In cyclic symmetry analysis, this command, or the HOUTPUT command, is required to obtain data recovery.

**NSM**

Selects Nonstructural Mass Set Entries

Selects nonstructural mass (NSM) set for mass generation.

**Format:**

NSM = n

**Example:**

NSM = 5

| Describer | Meaning  |
|-----------|--|
| n         | Set identification number of a nonstructural mass that appears on a NSM, NSML, NSM1, NSML1, or NSMADD Bulk Data entry (Integer > 0). |

**Remark:**

1. Different NSM sets may be selected for superelements and residuals but within a superelement or residual it may not change within the subcase structure.

**NVELOCITY****Velocity Normal Output Request**

Request output of velocity normal for structural grids in SOLs 108 and 111 only.

**Format:**

$$\text{NVELOCITY} \left[ \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix}, \text{ERP} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

$$\text{NVELOCITY} = \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix}$$

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as tabular listing of grid points for each excitation frequency (Default). |                   |                      |
| SORT2            | Output will be presented as tabular listing of excitation frequencies for each grid point.          |                   |                      |
| PRINT or (blank) | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT            | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |   |
|--------------|---|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAGE yields the same output.               |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is In degrees.  |
| ERP          | Flag to accept ERP set convention.  |
| ALL          | Velocity normal will be computed for all surface structural grid points.  |
| n            | Set identification of a previously defined set of grid points. Velocity Normal will be computed for the grid points in this set only. |
| NONE         | Velocity normal will not be processed.  |

**Remarks:**

1. NVELOCITY = NONE overrides an overall request.
2. The PLOT option is used if results are requested for post-processing but no printed or punched output is desired.
3. Even with NVELOCITY=all, Velocity Normal will be computed only for structural grids at the surface of the model. Fluid grids, if exist, are not included in NVELOCITY output.
4. With ERP flag in NVELOCITY command, structural grids of all ERPPNL will be considered as candidates for NVELOCITY output and subjected to further screening of NVELOCITY set.
5. For 3D model, ERPPNL points to SET3 with all PSOLID IDs in the model is the easiest way to compute NVELOCITY of structural surface GRIDs.
6. This Case Control command can be used in SOL 108 and SOL 111 only

**OFREQUENCY****Output Frequency Set**

Selects a set of frequencies for output requests.

**Format:**

$$OFREQUENCY = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

**Examples:**

OFREQUENCY=ALL  
OFREQUENCY=15

| Descriptor | Meaning   |
|------------|---|
| ALL        | Output for all frequencies will be computed.  |
| n          | Set identification of a previously appearing SET command. Output for frequencies closest to those given on this SET command will be output (Integer > 0). |

**Remarks:**

1. In real eigenvalue, buckling, and complex eigenvalue analyses, the OMODES Case Control command allows for an alternate way of selecting the modes to be output based on their mode numbers. In these cases, if both the OMODES and OFREQUENCY requests appear, the OMODES request takes precedence.
2. If this command is not specified in the Case Control Section (or, in the case of real eigenvalue, buckling, and complex eigenvalue analyses, if neither the OMODES nor the OFREQUENCY request is specified), then output will be generated for all frequencies.
3. The number of solutions selected will always be equal to the number of quantities in the selected set. The closest values are used.
4. In flutter analysis (SOL 145), the selected set refers to the imaginary part of the complex eigenvalues. The physical interpretation of this quantity depends on the method of flutter analysis as follows:
  - K- or KE-method: velocity (input units).
  - PK-method: frequency.
5. In aeroelastic response analysis (SOL 146) with RLOAD selection, the selected set refers to the frequency (cycles per unit time).
6. In complex eigenvalue analysis (SOLs 107 and 110), the selected set refers to the imaginary part of the complex eigenvalues.
7. If this command is specified in more than one subcase, then it is recommended that the first subcase contain OFREQ=ALL, and that subsequent subcases contain OFREQ = n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
OFREQ = ALL $ 0.0 through 0.5
SUBCASE 2
SET 10 = 0.0 0.1 0.3
OFREQ = 10
DISP = ALL
SUBCASE3
SET 20 = 0.4 0.5
OFREQ = 20
STRESS = ALL
```

**OIMPERFECT****Request Output of Imperfection**

Request output imperfection vectors in SOL 400. Imperfection vectors are coordinate variations applied on grid points to express geometric imperfection.

**Format:**

$$OIMPERFECT\left(\begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, GEOM\right) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

OIMPERFECT(GEOM) = 5

OIMPERFECT(PLOT)=ALL

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| GEOM | To output grid points with updated coordinates in punch file.   |
| ALL  | Imperfection vectors for all points will be output.   |
| n    | Set identification of a previously appearing SET command. Only imperfection vectors of points with identification numbers that appear on this SET command will be output. |
| NONE | Imperfection vectors for no points will be output.  |

**Remark:**

1. This command must be above all subcases.

**OLOAD****Applied Load Output Request**

Requests the form and type of applied load vector output.

Format:

$$\text{OLOAD} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{c} \text{PSDF}, \text{ATO}, \text{CRMS} \\ \text{or RALL} \end{array} \right], \\ \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{CID} \end{array} \right] \end{array} \right] = \begin{cases} \text{ALL} \\ n \\ \text{NONE} \end{cases}$$

Examples:

OLOAD=ALL  
 OLOAD (SORT1, PHASE)=5  
 OLOAD (SORT2, PRINT, PSDF, CRMS, RPUNCH=20  
 OLOAD (PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |

| Descriptor | Meaning   |
|------------|---|
| PSDF       | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 10. |
| ATOC       | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 10.        |
| CRMS       | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 10.            |
| RALL       | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control. See Remark 10.                                |
| RPRINT     | Writes random analysis results to the print file (Default).   |
| NORPRINT   | Disables the writing of random analysis results to the print file.  |
| RPUNCH     | Writes random analysis results to the punch file.   |
| CID        | Requests printing of output coordinate system ID to printed output file (.f06).   |
| ALL        | Applied loads for all points will be output. See Remarks 1. and 7.  |
| NONE       | Applied load for no points will be output.  |
| n          | Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output (Integer > 0).  |

**Remarks:**

1. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all requested points (zero and nonzero) to be output.
3. OLOAD=NONE overrides an overall output request.
4. In the statics superelement solution sequences, and in the dynamics solution sequences (SOLs 107 through 112, 118, 145, 146, and 200). OLOADs are available for superelements and the residual structure. Only externally applied loads are printed. Loads transmitted from upstream superelements are not printed. Transmitted loads can be obtained with GPFORCE requests.

In the nonlinear transient analysis solution sequences (SOLs 129 and 159), OLOADs are available only for residual structure points and include loads transmitted by upstream superelements.

5. In nonlinear analyses SOL 106 and 129, OLOAD output will not reflect changes due to follower forces. But SOL 400 OLOAD output reflects the follower force change correctly. Notes, in SOL 400, the first OLOAD output before starting nonlinear iteration process, only reflects total non-follower force without thermal load and follower force.
6. Loads generated by the SPCD Bulk Data entry do not appear in OLOAD output.
7. In SORT1 format, OLOADs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and it is zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
8. OLOAD results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
9. In static inertia relief analysis, the OLOAD output includes both the inertia loads and applied loads.
10. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
11. Note that the CID keyword affects only grid point related output, such as DISP, VELO, ACCE, OLOAD, SPCF and MPCF. In addition, CID the keyword needs to appear only once in a grid point related output request anywhere in the Case Control Section to turn on the printing algorithm.
12. OLOAD output is not available in SOL 600.

**OMODES****Output Modes Set**

Selects a set of modes for output requests.

Format:

$$OMODES = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

Examples:

OMODES = ALL  
OMODES = 20

| Descriptor | Meaning  |
|------------|--|
| ALL        | Output for all extracted modes will be computed (Default).   |
| n          | Set identification of a previously appearing SET command. Output for those extracted modes appearing on this SET command will be computed. |

Remarks:

1. This command is valid only in SOLs 103, 105, 107, 110, 111, 112, 200 and 400. It is ignored in all other analyses.
2. In contrast to the OFREQENCY Case Control command, which provides an alternate way of selecting the modes to be output based on their frequencies, the OMODES command allows mode selection based on integer mode ID. For example:

```
SUBCASE 10
...
SET 11 = 1,3,5,7
OMODES = 11
DISP = ALL
...
SUBCASE 20
...
SET 21 = 25., 28., 31.
OFREQ = 21
DISP = ALL
...
```

3. If both the OMODES and the OFREQENCY requests appear, the OMODES request takes precedence.
4. If neither the OMODES nor the OFREQENCY request is specified, output will be generated for all modes.
5. Note that the OMODES command has no effect on the number of modes computed. It only selects a subset of the computed modes for which output is to be generated.

6. In superelement analysis, the set definition, using an OMODES command, for an upstream superelement, will not be recognized unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OMODES default is ALL. Note that the program does not check to see if this condition is satisfied.
7. If OMODES is used in conjunction with a FATIGUE case control command, the OMODES request must be for contiguous modes from 1 through n and the FTGLOAD bulk data entries cannot invoke modes greater than n.
8. OMODES is intended to be specified above all subcases or in the first subcase. In SOLs 200 and 400, the first subcase would be the first subcase with ANALYSIS=MODES, MTRAN, MFREQ, MCEIG, SEAERO, and FLUTTER. See the MODES command to allow different output requests for individual modes.

**OTIME****Output Time Set**

Selects a set of times for output requests.

Format:

$$OTIME = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

Examples:

OTIME =ALL  
OTIME =15

| Descriptor | Meaning  |
|------------|--|
| ALL        | Output for all times will be computed.   |
| n          | Set identification number of a previously appearing SET command. Output for times closest to those given on this SET command will be computed (Integer > 0). |

Remarks:

1. If the OTIME command is not supplied in the Case Control Section, then output for all times will be computed.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses only at peak times, etc.).
3. This command can be used in conjunction with the MODACC module to limit the times for which modal acceleration computations are performed.
4. If this command is specified in more than one subcase in the modal solution sequences, then it is recommended that the first subcase contain OTIME=ALL, and that subsequent subcases contain OTIME=n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
  OTIME ALL
SUBCASE 2
  OTIME = 10
  SET 10 = . . .
  DISP = ALL
SUBCASE 3
  OTIME = 20
  SET 20 = . . .
  STRESS = ALL
```

5. The OTIME command is not effective in nonlinear transient analysis (SOL 129). However, the OTIME command can be used in the nonlinear transient thermal analysis (SOL 159) to limit the output to specified output times.

6. In superelement analysis, the set definition, using an OTIME command for an upstream superelement, will not be recognized unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OTIME default is ALL. Note that the program does not check to see if this condition is satisfied.

**OUTPUT****Case Control Delimiter**

Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.

Format:

$$OUTPUT \left[ \begin{array}{c} PLOT \\ POST \\ XYOUT \\ XYPILOT \\ CARDS \end{array} \right]$$

Examples:

OUTPUT (POST)  
OUTPUT (PLOT)  
OUTPUT (XYOUT)

| Descriptor       | Meaning   |
|------------------|---|
| PLOT             | Beginning of the structure plotter request. This command must precede all structure plotter control commands. Plotter commands are described in <a href="#">OUTPUT(PLOT) Commands, 743</a> .                                      |
| POST             | Beginning of grid point stress SURFACE and VOLUME commands. This command must precede all SURFACE and VOLUME commands. These commands are described in <a href="#">OUTPUT(PLOT) Commands, 743</a> .                               |
| XYOUT or XYPILOT | Beginning of curve plotter request. This command must precede all curve plotter control commands. XYPILOT and XYOUT are equivalent. Curve plotter commands are described in <a href="#">X-Y PLOT Commands, 662</a> .              |
| CARDS            | The OUTPUT(CARDS) packet is used by the MSGSTRESS program. See the <i>MSGMESH Analyst's Guide</i> for details. These commands have no format rules. This package must terminate with the command ENDCARDS (starting in column 1). |

Remarks:

1. The structure plotter request OUTPUT(PLOT), the curve plotter request OUTPUT(XYOUT or XYPILOT), and the grid point stress requests (OUTPUT(POST)) must follow the standard Case Control commands.
2. If OUTPUT is specified without a descriptor, then the subsequent commands are standard Case Control commands.
3. Case Control commands specified after OUTPUT(POST) are SURFACE and VOLUME.

**OUTRCV****P-element Output Option Selection**

Selects the output options for the p-elements defined on an OUTRCV Bulk Data entry.

**Format:**

OUTRCV=n

**Examples:**

OUTRCV=10

OUTR=25

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of a OUTRCV Bulk Data entry (Integer > 0). |

**Remark:**

1. The OUTRCV command is optional. By default, p-element output uses the defaults specified for CID and VIEW as described in the OUTRCV Bulk Data entry description.

**P2G****Direct Input Load Matrix Selection**

Selects direct input load matrices.

**Format:**

P2G=name

**Example:**

```
P2G = LDMIG
P2G = LDMIG1, LDMIG2, LDMIG3
SET 100 = LDMIG, L1, L8
P2G = 100
P2G = 1.25*LDMIG1, 1.0*LDMIG2, 0.82*LDMIG3
```

| Descriptor | Meaning  |
|------------|--|
| name       | Name of a $[P_g^2]$ matrix to be input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 4. (Character). |

**Remarks:**

1. Terms are added to the load matrix before any constraints are applied.
2. The matrix must be columnar in form (e.g., Field 4 on DMIG entry, IFO, must contain the integer 9.)
3. A scale factor may be applied to this input using the user parameter PARAM,CP2. See [Parameters, 783](#).
4. The formats of the name list:
  - a. Names without factor.  
Names separated by a comma or blank.
  - b. Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by a comma or blank. The factors are real numbers. Each name must be paired with a factor including 1.0.
5. SOL 101: P2G should be selected above all subcase. The number of columns specified for NCOL on the DMIG Bulk Data entry must equal the number of subcases.  
SOL 106 and SOL 400: There are two choices. P2G may be selected in every subcase (or step in SOL 400), with NCOL=1. Otherwise, P2G may be selected above the subcases and PARAM,CP2 selected in every subcase.

**PACCELERATION****Particle Acceleration Output Request for SOL 108/111**

Request the form and frequency steps of particle acceleration output.

**Format:**

$$PACCELERATION \left( \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix}, \begin{bmatrix} SOLUTION = \{ \text{ALL} \\ setf \} \end{bmatrix} \right) = \left\{ \begin{array}{l} ALL \\ setg \\ NONE \end{array} \right\}$$

**Example:**

```
PACCELERATION = ALL
SET 20 = 104 THRU 204, 1005 THRU 1901
SET 50 = 105.0, 250.0, 310.0
PACCE (PUNCH, SOLUTION=50) = 20
```

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary). Use of either REAL or IMAG yields the same output.   |
| PHASE        | Requests polar format (magnitude and phase). Phase output is in degrees.   |
| SOLUTION     | Selects a set of excitation frequencies for which the particle accelerations will be processed. The default is all excitation frequencies. See Remark 3. |
| setf         | Set identification of excitation frequencies.  |
| setg         | Set identification of grid points on wetted surface. See Remark 2.   |

**Remarks:**

1. This entry will be available only for fluid-structure coupling problem. The particle acceleration is input force vector to each domain via wetted surface in frequency response analysis.
2. Both fluid and structure grid points can be selected. The particle accelerations of the grid points not on wetted surface will be zero.

3. The selected frequency must be part of the excitation frequencies. If not, the nearest excitation frequency will be selected.
4. Only SORT1 form is supported.

**PAGE**

**Page Eject**

Causes a page eject in the echo of the Case Control Section.

**Format:**

PAGE

**Example:**

PAGE

**Remarks:**

1. PAGE appears in the printed echo prior to the page eject.
2. PAGE is used to control paging in large Case Control Sections.

**PARAM** Parameter Specification

Specifies values for parameters. Parameters are described in [Parameters, 783](#).

**Format:**

PARAM,n,V1,V2

**Examples:**

PARAM, GRDPNT, 0  
PARAM, K6ROT, 1.0

| Descriptor | Meaning  |
|------------|--|
| n          | Parameter name (one to eight alphanumeric characters, the first of which is alphabetic). |
| V1, V2     | Parameter value based on parameter type, as follows:                                     |

| Type                      | V1                     | V2                     |
|---------------------------|------------------------|------------------------|
| Integer                   | Integer                | Blank                  |
| Real, single precision    | Real                   | Blank                  |
| Character                 | Character              | Blank                  |
| Real, double precision    | Real, Double Precision | Blank                  |
| Complex, single precision | Real or Blank          | Real or Blank          |
| Complex, double precision | Real, Double Precision | Real, Double Precision |

**Remarks:**

1. The PARAM command is normally used in the Bulk Data Section and is described in the [Bulk Data Entries, 1117](#).
2. The parameter values that may be defined in the Case Control Section are described in [Parameters, 783](#). Case Control PARAM commands in user-written DMAPs requires the use of the PVT module, described in the [MSC Nastran DMAP Programmer's Guide](#).

**PARTN**

## Partitioning Vector Specifications

Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17).

**Format:**

PARTN=n

**Example:**

PARTN=10

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of a previously appearing SET command (Integer > 0). |

**Remarks:**

1. The PARTN command and the DMAP module MATMOD provide a convenient method for building a partitioning vector for use in DMAP modules such as PARTN and MERGE.
2. The PARTN command is no longer applicable to coupled fluid-structure analysis. It has been replaced by the FSLPOUT command.

**PEAKOUT**

Control peaks identification process for frequency response analysis

Format:

$$\text{PEAKOUT } \text{NPEAK} = P, \text{NEAR} = q, \text{LFREQ} = r, \text{HFREQ} = s, \text{RTYPE} = \begin{cases} \text{DISP} \\ \text{VELO} \\ \text{ACCE} \end{cases}, \text{PSCALE} = \begin{cases} \text{NONE} \\ \text{DB} \\ \text{DBA} \end{cases}$$

Example:

```
PEAKOUT NPEAK=4, NEAR=2.0, LFREQ=10.0, HFREQ=200.0, RTYPE=DISP,
PSCALE=DBA
```

| Descriptor | Meaning   |
|------------|---|
| NPEAK      | Requests the desired number of peaks to extract. See remark 2. (Integer, Default=5).  |
| NEAR       | Minimum allowed frequency between two peaks. If two peaks are closer than this value, the lower frequency peak will be ignored. (Real, Default=0.01 Hz) |
| LFREQ      | Lowest frequency used in peak identification. (Real, Default=lowest forcing frequency or 0.0 Hz)  |
| HFREQ      | Highest frequency used in peak identification. (Real, Default=highest forcing frequency or 1.0e10 Hz)   |
| RTYPE      | Results type for peak identification in structural domain. (Default=DISP)   |
| PSCALE     | Scaling method for acoustic pressure results in fluid domain. See remark 3. (Default=NONE)  |

Remarks:

1. This command is meaningful only in frequency response analysis (SOLs 108 and 111) and SOL 200 with DFREQ and MFREQ. It is ignored in all other analyses.
2. The actual number of peaks found may be fewer than the desired number.
3. When requesting dB or dBA, the parameter PREFDB is used as a reference pressure for dB calculation, see ACOUT parameter.
4. All GRIDs referenced on any setdof of PFMODE, PFGRID and PFPANEL requests are required (but not limited to) on a DISP output command. The DISP command must be placed above the SUBCASE level.
5. If non sparse data recovery is requested using PARAM,SPARSEDR,NO, then also PARAM,DDRMM,-1 is required. Sparse Data recovery is default.
6. External super elements are not supported

**PFGRID**

## Acoustic Grid Participation Factor Output Request

Requests the form and type of acoustic grid participation factor output.

Format:

$$\text{PFGRID} \left[ \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right), \left( \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right), \left[ \text{GRIDS} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setg} \end{array} \right\} \right], \right. \\ \left. \left[ \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \overline{\text{setf}} \\ \text{PEAK} \\ \text{NONE} \end{array} \right\} \right] \right) = \left\{ \begin{array}{c} \text{setdof} \\ \text{NONE} \end{array} \right\}$$

Example:

```
SET 10 = 11217
SET 20 = 25., 30., 35.
PFGRID(PHASE, SOLUTION=20) = 10
SET 40 = 11217
PEAKOUT NPEAK=4, NEAR=2.0,
PFGRID(PLOT, PHASE, SOLUTION=PEAK) = 40
```

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Real and imaginary part of complex results will be output (Default).                                     |
| PHASE        | Magnitude and phase of complex results will be output.   |
| GRIDS        | Keyword selecting the structural grid points to be processed; the default is all structural grid points. |
| setg         | Identifier of a set containing the identifiers of the structural grid points to be processed.            |

| Descriptor | Meaning   |
|------------|---|
| SOLUTION   | Keyword selecting a set of excitation frequencies for which the participation factors will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters. |
| setf       | Identifier of a set of excitation frequencies.  |
| setdof     | Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.  |

Remarks:

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of fluid degrees of freedom.
2. Acoustic grid participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.

**PFMODE**

## Modal Participation Factor (MPF) Output Request

Requests the form and type of modal participation factor (MPF) output.

Format:

$$\begin{aligned}
 & \text{PFMODE} \left[ \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right), \left( \begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right) [\text{PRTMSG}], \right. \\
 & [\text{SORT} = \text{sorttype}], [\text{KEY} = \text{sortitem}], \left[ \text{ITEMS} = \left\{ \begin{array}{c} \text{ALL} \\ (\text{itemlist}) \end{array} \right\} \right], \\
 & \left[ \text{FLUIDMP} = \left\{ \begin{array}{c} \text{ALL} \\ m_f \\ \text{NONE} \end{array} \right\} \right], \left[ \text{STRUCTMP} = \left\{ \begin{array}{c} \text{ALL} \\ m_s \\ \text{NONE} \end{array} \right\} \right], \\
 & \left[ \text{PANELMP} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setp} \\ \text{NONE} \end{array} \right\} \right], \left[ \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setf} \\ \text{PEAK} \\ \text{NONE} \end{array} \right\} \right], [\text{FILTER} = \text{fratio}], \\
 & [\text{NULL} = \text{ipower}])] = \left\{ \begin{array}{c} \text{setdof} \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

Examples:

```

SET 20 = 11/T3, 33/T3, 55/T3
SET 30 = 420., 640., 660.
PFMODE (STRUCTURE, SOLUTION = 30, FILTER = 0.01, SORT = ABSD) = 20
SET 40 = 1222, 1223
SET 50 = 10., 12.
PFMODE (FLUID, STRUCTMP=ALL, PANELMP=ALL, SOLUTION=50, SORT=ABSD) = 40
PEAKOUT NPEAK=4, NEAR=2.0,
PFMODE (FLUID, STRUCTMP=ALL, PANELMP=ALL, SOLUTION=PEAK) = 40

```

| Descriptor | Meaning   |
|------------|---|
| STRUCTURE  | Requests output of MPFs for the response of structural degrees of freedom. See Remark 8. (Default).               |
| FLUID      | Requests output of MPFs for the response of acoustic grid points (one degree of freedom per point). See Remark 9. |

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|          |  |
|----------|--|
| PRTMSG   | Request generation of diagnostic messages for badly-defined output sets during PF computations. These badly defined sets generally produce no output. See Remark 15.   |
| SORT     | Keyword selecting the sort type. Default is sorting by increasing natural mode number. See Remark 7.   |
| sorttype | Sort option:<br><br>ABSA    output will be sorted by absolute value in ascending order.<br><br>ABSD    output will be sorted by absolute value in descending order.<br><br>ALGA    output will be sorted by algebraic value in ascending order.<br><br>ALGD    output will be sorted by algebraic value in descending order. |
| KEY      | Keyword selecting the output item to be used for sorting; default is FRACTION. See Remark 7.   |
| sortitem | Item from the item list, see the following table, on which the sort operation is performed.  |
| ITEMS    | Keyword specifying data selected for output to the .pch file   |
| itemlist | One or more of the items in the following table:   |

Table 5-1 Item List Table

| Item Identifier | Description   |
|-----------------|---|
| RESPONSE        | Modal participation factor.   |
| PROJECTION      | Projected modal participation factor.   |
| FRACTION        | Normalized projected modal participation factor.  |
| SCALED          | Projected modal participation factor divided by largest magnitude of all modal participation factors. |
| MODEDISP        | Real and imaginary part of modal participation factors. See Remark 7.                                 |
| MODERESP        | Magnitude and phase relative to total response of modal participation factors. See Remark 7.          |

|          |   |
|----------|---|
|          | If more than one item is selected, the list must be enclosed in parentheses.  |
| FLUIDMP  | Keyword to select output of fluid MPFs. See Remarks 2. and 9.   |
| $m_f$    | Number of lowest fluid modes for which MPFs will be computed. See Remark 9.   |
| STRUCTMP | Keyword to select output of structural MPFs. See Remarks 8. and 9.  |
| $m_s$    | Number of lowest structural modes for which MPFs will be computed. See Remarks 8. and 9.  |
| PANELMP  | Keyword to select output of panel MPFs. See Remark 2.   |
| setp     | Identifier of a set of panels.  |
| SOLUTION | Selects a set of excitation frequencies for which MPFs will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters. See Remark 10. |
| setf     | Identifier of a set of excitation frequencies. See Remark 10.   |
| FILTER   | Keyword specifying the value of a filter to be applied to the printed output. See Remark 11.  |
| fratio   | Filter value (Default is 0.001). See Remark 11.   |
| NULL     | Keyword specifying the power of ten used to detect a null response. See Remark 12.  |
| ipower   | Power of ten used to detect a null response (Default is 12). See Remark 12.   |
| setdof   | Identifier of a set of structural degrees of freedom or acoustic grid points (one degree of freedom per point) for which MPFs are to be processed. See Remark 3.  |

**Remarks:**

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of acoustic grid points.
2. Keywords FLUIDMP and PANELMP are only valid if FLUID is specified.
3. If STRUCTURE is specified, *setdof* must reference a set of structural degrees of freedom. If FLUID is specified, *setdof* must reference a set of acoustic grid points.
4. Acoustic MPFs are available in a coupled modal frequency response analysis (SOL 111) only.
5. Printed output includes results for ALL the data items described in the Item List Table.
6. Punched output includes results for only the data items selected by the ITEMS keyword.
7. MPFs are sorted by increasing order of mode number unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.
8. The STRUCTURE option selects MPF calculations for structural degrees of freedom. The STRUCTMP value defines the number of lowest structural modes used in final output preparation prior to any filtering. The default is ALL. If STRUCTMP=NONE, no structural MPF is generated.
9. The FLUID option selects MPF calculations for acoustic grid points (one degree of freedom per point). The STRUCTMP value defines the number of lowest structural modes used in final output preparation of acoustic structure MPFs prior to any filtering. The default is NONE. The FLUIDMP value defines the number of lowest fluid modes used in final output preparation of acoustic fluid MPFs prior to any filtering. The default is NONE.
10. The SOLUTION keyword can be used to select a subset of solutions available. If *setf* is specified, the items in the set are excitation frequencies.
11. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
12. If the magnitude of the total response at a selected response degree of freedom is less than  $10^{-ipowr}$ , then no MPFs are processed. If *ipower* is not in the range of 1 to 31, the default of 12 is used.
13. Acoustic panel MPFs are normalized using the panel response instead of the total response.
14. If present, output of acoustic structural MPFs includes the load participation factor. The load participation factor has a mode number of 0 and a resonance frequency of 0.
15. Output sets that define no valid model degrees of freedom for data recovery produce no PF output. This condition is usually discovered prior to actual computations and an informational message is generated. Occasionally, when multiple subcases are involved and one of the subcases references an invalid set, the PF calculations are still performed for any valid combinations. PRTMSG can be used to aid in identification of invalid set data under these conditions.

**PFPANEL****Acoustic Panel Participation Factor Output Request**

Requests the form and type of acoustic panel participation factor output.

**Format:**

$$\begin{aligned}
 PFPANEL & \left[ \left[ \begin{array}{c} FLUID \\ STRUCTURE \end{array} \right] \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right] [PRTMSG], \left[ \begin{array}{c} REAL \text{ or } IMAG \\ PHASE \end{array} \right], \left[ \begin{array}{c} PANEL = \left\{ \begin{array}{c} ALL \\ setp \end{array} \right\} \end{array} \right] \right. \\
 & \left. [SORT = sorttype], [KEY = sortitem], \left[ \begin{array}{c} ITEMS = \left\{ \begin{array}{c} ALL \\ (itemlist) \end{array} \right\} \end{array} \right], \right. \\
 & \left. \left[ \begin{array}{c} SOLUTION = \left\{ \begin{array}{c} ALL \\ setf \\ PEAK \\ NONE \end{array} \right\} \end{array} \right], [FILTER = fratio], [NULL = ipower] \right) \\
 & = \left\{ \begin{array}{c} setdof \\ NONE \end{array} \right\}
 \end{aligned}$$

**Example:**

```

SET 10 = 10., 12.
SET 20 = 1222, 1223
PFPANEL (SOLUTION=10, FILTER=0.01, SORT=ABSD) = 20
SET 30 = 5001/T2, 6502/T3
PFPANEL (STRUCTURE, SOLUTION=10) = 30
SET 40 = 11217
PEAKOUT NPEAK=4, NEAR=2.0,
PFPANEL (SOLUTION=PEAK, FILTER=0.01, SORT=ABSD) = 40
  
```

| Descriptor | Meaning   |
|------------|---|
| FLUID      | Request output of MPFs for the response of acoustic grid points (one degree of freedom per point). See Remark 11. (Default) |
| STRUCTURE  | Request output of MPFs for the response of structure degrees of freedom. See Remark 12.                                     |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

| Descriptor   | Meaning   |
|--------------|---|
|              | * The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated. |
| PRTMSG       | Request generation of diagnostic messages for badly-defined output sets during PF computations. These badly defined sets generally produce no output. See Remark 12.  |
| REAL or IMAG | Real and imaginary part of complex results will be output (Default).  |
| PHASE        | Magnitude and phase of complex results will be output.  |
| PANEL        | Keyword to select the panels to be processed; default is all panels.  |
| setp         | Identifier of a set of panels.  |
| SORT         | Keyword selecting the sort type. Default is alphabetic sorting by panel name.   |
| sorttype     | Sort option:  |
| ABSA         | Output will be sorted by absolute value in ascending order.   |
| ABSD         | Output will be sorted by absolute value in descending order.  |
| ALGA         | Output will be sorted by algebraic value in ascending order.  |
| ALGD         | Output will be sorted by algebraic value in descending order.   |
| KEY          | Keyword selecting the output item to be used for sorting; default is FRACTION.  |
| sortitem     | Item from the item list, see the following table, on which the sort operation is performed.   |
| ITEMS        | Keyword specifying data selected for output to the .pch file.   |
| itemlist     | One or more of the items in the following table:  |

Table 5-2 Item List Table

| Item Identifier | Description   |
|-----------------|---|
| RESPONSE        | Modal participation factor.   |
| PROJECTION      | Projected modal participation factor.   |
| FRACTION        | Normalized projected modal participation factor.  |
| SCALED          | Projected modal participation factor divided by largest magnitude of all modal participation factors. |
| MODEDISP        | Real and imaginary part of modal participation factors.   |
| MODERESP        | Magnitude and phase relative to total response of modal participation factors.                        |

If more than one item is selected, the list must be enclosed in parentheses.

|          |   |
|----------|---|
| SOLUTION | Keyword selecting a set of excitation frequencies for which the participation factors will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters. |
| setf     | Identifier of a set of excitation frequencies.  |
| FILTER   | Keyword specifying the value of a filter to be applied to the printed output.   |
| fratio   | Filter value (Default is 0.001), see Remark 7.  |
| NULL     | Keyword specifying the power of ten used to detect a null response, see Remark 8.   |
| ipower   | Power of ten used to detect a null response (Default is 12), see Remark 8.  |
| setdof   | Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.  |

#### Remarks:

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of fluid degrees of freedom.
2. Acoustic panel participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. Printed output includes results for all the data items described in the itemlist table.
4. Punched output includes results for only the data items selected by the ITEMS keyword.
5. Panel participation factors are alphabetically sorted by panel names unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.

6. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.
7. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
8. If the magnitude of the total response at a selected response degree of freedom is less than  $10^{-\text{ipowr}}$ , then no modal participation factors are processed. If ipower is not in the range of 1 to 31, the default of 12 is used.
9. If present, output includes the load participation factor. The panel name of the load participation factors is -LOAD-.
10. The FLUID option selects panel PF calculation for acoustic grid points (one degree of freedom per point) and setdof should be identification numbers.
11. The STRUCTURE option selects panel PF calculation for structure grid points and setdof should be identification numbers and component codes.
12. Output sets that define no valid model degrees of freedom for data recovery produce no PF output. This condition is usually discovered prior to actual computations and an informational is generated. Occasionally, when multiple subcases are involved and one of the subcases references an invalid set, the PF calculations are still performed for any valid combinations. PRTMSG can be used to aid in identification of invalid set data under these conditions.

**PLOTID****Plotter Identification**

Defines a character string that will appear on the first frame of any plotter output.

**Format:**

PLOTID=title

**Example:**

PLOTID=BLDG. 125 BOX 91

| Descriptor | Meaning               |
|------------|-----------------------|
| title      | Any character string. |

**Remarks:**

1. PLOTID must appear before the OUTPUT(PLOT) or OUTPUT(XYOUT) Case Control commands.
2. The presence of PLOTID causes a special header frame to be plotted, with the supplied identification plotted several times. The header frame allows plotter output to be identified easily.
3. If no PLOTID command appears, no ID frame will be plotted.
4. The PLOTID header frame will not be generated for the table plotters.

**POST**

## Postprocessor Data Specifications

Controls selection of data to be output for postprocessing functions via the OUTPUT2 module interface for selected commercial postprocessor products.

**Format:**

$$\text{POST} \left\{ \begin{array}{l} \text{TOFILE} \\ \text{TOCASE} \end{array} \right\} \left\{ \begin{array}{l} \text{furn} \\ \text{filename} \end{array} \right\} [\text{ppname}] [\text{oplist}]$$

**Examples:**

```
POST PATRAN TOFILE 51 NOSTRESS
POST TOFILE SUBCASE8
POST TOCASE SUFNAME1
```

| Descriptor | Meaning   |
|------------|---|
| TOFILE     | Keyword to specify the destiny of .op2 output files (No default if it appears above all subcases).                                  |
| TOCASE     | Keyword to specify the destiny of subcase results to user-defined .f06 output files. (No default if it appears above all subcases.) |
| furn       | Fortran file unit reference number where data will be written (Integer > 0).  |
| filename   | Suffix filename (see Remark 8., 9. and 11.) (Char8)   |
| ppname     | Name of the target post-processor program for TOFILE option (Default = PATRAN).   |
| oplist     | Names of output items to be processed.  |

**Remarks:**

1. The POST Case Control command controls the placement of output data on external FORTRAN files for use by commercial postprocessors. Use of the POST command generates the proper value for the POST DMAP parameter associated with the particular postprocessor. All of the other parameter controls related to the POST DMAP parameter remain in effect, and are described in [Parameters, 783](#). The products supported are identified in the following table. PATRAN is the default postprocessor name used for ppname. DBC output (POST=0) cannot be controlled by the POST command.

| ppname     | Product         | PARAM,POST,Value |
|------------|-----------------|------------------|
| PATRAN     | MSC Patran V3   | -1               |
| SDRC       | SDRC I-DEAS     | -2               |
| NF         | MSC/LMS NF      | -4               |
| FEMTOOLS   | DDS/FemTools    | -5               |
| UNIGRAHICS | EDS/Unigraphics | -6               |

2. The TOFILE descriptor is followed by the specification of either a FORTRAN unit reference number, or a file name associated with the external file that receives the output data. If a FORTRAN unit number is used, the file must be associated with it via the ASSIGN File Management Statement. If POST appears above all subcases, TOFILE must be used to specify either a FORTRAN unit reference number or a file name. The default value of TOFILE, which appears under a subcase, will inherit from the value given in the POST above all subcases. If the unit reference number is associated with a form=formatted file, changes in unit numbers across subcases are not allowed.
3. The data that can be controlled for each postprocessor product is limited, and is identified under the description of the POST and related DMAP parameters as described in [Parameters, 783](#). The keywords that can be used for the oplist options are shown in the following table. If an output item supported by a particular postprocessor is described in [Parameters, 783](#) but is not listed here, then the POST command cannot be used to control its output to the external file.

| Output Item                          | oplist Keyword | Case Command |
|--------------------------------------|----------------|--------------|
| Displacements                        | [NO]DISPLACE   | DISP         |
| Forces of single point constraint    | [NO]SPCFORCE   | SPCFORCE     |
| Element forces                       | [NO]FORCES     | ELFO/FORCE   |
| Element stresses                     | [NO]STRESS     | ELST/STRESS  |
| Element strain energy                | [NO]ESE        | ESE          |
| Grid point force balance             | [NO]GPFORCE    | GPFORCE      |
| Stress at grid points                | [NO]GPSIGMA    | STRESS       |
| Strain/curvature at grid points      | [NO]GPEPSILON  | STRAIN       |
| Composite element failure indices    | [NO]PLYFAILURE | STRESS       |
| Element kinetic energy               | [NO]EKE        | EKE          |
| Element energy loss                  | [NO]EDE        | EDE          |
| Multi-point constraint forces        | [NO]MPCFORCE   | MPCFORCE     |
| Composite lamina stresses            | [NO]PLYSIGMA   | STRESS       |
| Composite lamina strains             | [NO]PLYEPSILON | STRAIN       |
| Element strains                      | [NO]STRAIN     | STRAIN       |
| Grid point stresses                  | [NO]GPSTRESS   | GPSTRESS     |
| Grid point strains                   | [NO]GPSTRAIN   | GPSTRAIN     |
| Applied loads                        | [NO]LOAD       | OLOAD        |
| No items to be output                | NONE           | -----        |
| Structure mode participation factors | [NO]SMPF       | PFMODE       |

4. Output data items must have been generated via the appropriate Case Control command for the data to be available for postprocessing options. For example, the specification of SPCF in the oplist of the POST command will not produce forces of single point constraint on the POST output file unless there is a SPCF Case Control command present. Refer to the tables under the POST parameter description in [Parameters, 783](#) for a list of the output items supported by each postprocessor.
5. Any data generated by a Case Control command is automatically included in the oplist of the POST command. If output data is not wanted for a particular case, then the characters “NO” should be the first two characters of the keyword in the oplist. For example, NODISP specifies that displacements are not to be posted to the output file, even though they have been requested via the DISP Case Control command. Alternatively, the related POST parameters may be used. For example, to avoid outputting any displacements whatsoever to the .op2 file, use a PARAM, OUG, NO Bulk Data entry.
6. Certain data (e.g., geometry) is always generated and is not dependent upon the presence of a Case Control command in the input data. The POST command affects the placement of this data on the external file only insofar as the selection of the postprocessor defines the value of the POST DMAP parameter value. The actions described in [Parameters, 783](#) under the POST parameter description will prevail for the particular value of POST associated with the selected postprocessor. The primary purpose of the POST command is to give the user more control over subcase-dependent output data being stored on the external OUTPUT2 file.
7. If a POST command is present within any subcase, a POST command must also be present above the subcase level. The placement of the POST command above the subcase level causes a cumulative effect on POST commands in subsequent subcases. Any options specified above the subcase level propagate down into the POST command within a subsequent subcase. Thus, if a POST command specifies NODISP (no displacement output wanted) above the subcase level, then a POST command with the DISP option would be required within a subcase to generate any output to the OUTPUT2 file for displacements. This also implies that changing the OUTPUT2 file unit reference number with the TOFILE option in a subcase causes all output quantities currently scheduled for output to be switched to the new unit number, not just those in the oplist for the current POST command.
8. When the name of an output file is specified by keyword TOFILE, the ASSIGN statement in the File Management Section (FMS) can be used to specify the full path of its root name. The logical-keyword for the root name is OUTPUT2F. The default root name is the Nastran job name. FORTRAN unit reference number 19 has been reserved by Nastran for OUTPUT2F, although the user can assign other FORTRAN unit numbers to it. The full file name is in the form of <root name>.<suffix filename>.
9. When the name of an output file is specified by keyword TOCASE, the ASSIGN statement in the File Management Section can be used to specify the full path of its root name. The logical-keyword for the root name is OPCASE. The default root name is the Nastran job name. FORTRAN unit reference number 22 has been reserved by Nastran for OPCASE, although the user can assign other FORTRAN unit numbers to it. The full file name is in the form of <root name>.<suffix filename>. Also, ppname and oplist are not required. If ppname and oplist are specified, they will be ignored. Suffix filename must be specified with keyword TOCASE.
10. POST commands using TOCASE for structure mode participation factor output (SMPF) are not supported and will be ignored.

11. The TOFILE option lets you control the output results (per the table in Remark 3) that are placed on the specified file in OP2 format. The "POST TOFILE SUBASE8" example directs output to the file named <root name>.subcase8.op2. The TOCASE option directs standard printed output requested for a subcase to the specified file in .f06 format. The "POST TOCASE SUFNAM1" example directs output to the file named <root name>.sufnam1.f06. The TOCASE and TOFILE describers are independent of one another. POST commands using each describer may be present in a subcase to specify different output destinations for both .op2 and .f06 outputs at the same time. For both the TOCASE and TOFILE options, the resulting file name will not have the .op2 or the .f06 file extension automatically appended if the file suffix name contains a period (".").
12. The TOCASE keyword is not supported for printout in the f06 file coming from X-Y Plot commands.

## PRESSURE

### Pressure Output Request

Requests form and type of pressure output. Analogous to the DISPLACEMENT Case Control command.  
See the description of the DISPLACEMENT Case Control command, [DISPLACEMENT \(Case\), 298](#).

**RANDOM****Random Analysis Set Selection**

Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.

**Format:**

$$\text{RANDOM} = \left\{ \begin{array}{c} n \\ i \end{array} \right\}$$

**Examples:**

RANDOM=177

SET 10=100 110 120

RANDOM=10

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification number of a previously appearing SET command, which in turn references multiple RANDPS/RANDT1 Bulk Data entries with different set identification numbers. |
| i          | Set identification number of RANDPS and RANDT1 Bulk Data entries to be used in random analysis (Integer > 0).   |

**Remarks:**

1. RANDOM must select RANDPS Bulk Data entries to perform random analysis.
2. RANDOM must appear in the first subcase of the current loop. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command, or changes in the K2PP, M2PP, or B2PP commands.
3. If RANDPS entries are used in a superelement analysis, the RANDOM command may be specified above the subcase level if a condensed subcase structure (SUPER=ALL) is used. If a condensed subcase structure is not used, then a unique RANDOM selection of a unique RANDPS entry must be specified within each of the desired superelement subcases.
4. P-elements are not supported in random analysis.
5. If a SET is referenced by n, then the SET identification number must be unique with respect to all RANDPS/RANDT1 set identification numbers.

**RCROSS**

## Cross-Power Spectral Density and Cross-Correlation Function Output Request

Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.

Format:

$$\text{RCROSS} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] \left[ \begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], [\text{PUNCH}], [\text{PSDF}, \text{CORF}, \text{RALL}] \end{array} \right] = n$$

**Example:**

```
RCROSS (PHASE, PSDF, CORF) = 10
RCROSS (RALL, NOPRINT, PUNCH) = 20
RCROSS = 30
```

| Descriptor   | Meaning  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output for cross-power spectral density function. Use of either REAL or IMAG yields the same output (Default). |
| PHASE        | Requests polar format (magnitude and phase) of complex output for cross-power spectral density function. Phase output is in degrees.                                       |
| PRINT        | Write output to print file (Default).  |
| NOPRINT      | Do not write output to print file.   |
| PUNCH        | Write output to punch file.  |
| PSDF         | Requests the cross-power spectral density function be calculated and output for random analysis postprocessing (Default).  |
| CORF         | Requests the cross-correlation function be calculated and output for random analysis postprocessing.   |
| RALL         | Requests both the cross-power spectral density function and cross-correlation function be calculated and output for random analysis postprocessing.                        |
| n            | Identification number of the RCROSS Bulk Data entry to be used in random analysis (Integer > 0).   |

**Remarks:**

1. Case Control command RCROSS must be used along with Case Control command RANDOM. See Remarks under the [RANDOM \(Case\), 547](#) Case Control command description.

2. Response quantities such as DISPLACEMENT, STRESS, and FORCE must be requested by corresponding Case Control commands in order to compute cross-power spectral density and cross-correlation functions between the two response quantities specified by the RCROSS Bulk Data entry. It is recommended that those requests be put above the subcase level to avoid the situation that some response quantities are missing when it comes to the random analysis postprocessing.
3. The response quantities must belong to the same superelement. The cross-power spectral density and cross-correlation functions between the two responses, which belong to the different superelements, are not supported.

**REPCASE****Repeat Output Subcase Delimiter**

Delimits and identifies a repeated output subcase.

**Format:**

REPCASE=n

**Example:**

REPCASE=137

| Descriptor | Meaning                                      |
|------------|--|
| n          | Subcase identification number. (Integer > 1) |

**Remarks:**

1. n must be strictly increasing (i.e., must be greater than all previous subcase identification numbers).
2. REPCASE defines a subcase that is used to make additional output requests for the previous real subcase. This command is required because multiple output requests for the same item are not permitted in the same subcase.
3. One or more repeated subcases (REPCASEs) must immediately follow the subcase (SUBCASE) to which they refer.
4. REPCASE may be used only in statics and normal modes analysis. In normal modes analysis, output for only one mode is obtained. This output corresponds to the SUBCASE preceding the REPCASE.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the REPCASE with a TEMP(LOAD) Case Control command, or the element deformation state with a DEFORM command.
6. Repcase is not applicable to SOLs 400, 600, and 700.

**RESVEC**

Controls Residual Vectors

Specifies options for and calculation of residual vectors.

Format:

$$RESVEC \left[ \left[ \begin{matrix} INRLOD \\ NOINRL \end{matrix} \right], \left[ \begin{matrix} APPLOD \\ NOAPPL \end{matrix} \right], \left[ \begin{matrix} ADJLOD \\ NOADJLOD \end{matrix} \right] \left[ \begin{matrix} RVDOF \\ NORVDO \end{matrix} \right], \left[ \begin{matrix} DAMPLOD \\ NODAMP \end{matrix} \right], \left[ \begin{matrix} DYNRSP \\ NODYNRSP \end{matrix} \right] \right] =$$

$$\left\{ \begin{array}{l} SYSTEM/NOSYSTEM \\ COMPONENT/NOCOMPONENT \\ BOTH \text{ or } YES \\ NO \end{array} \right\}$$

$$RESVEC(FLUID) = \{ YES | NO \}$$

Examples:

RESVEC=SYSTEM  
 RESVEC (NOINRL)=COMPONENT  
 RESVEC=NO  
 RESVEC (FLUID)=NO

| Descriptor                | Meaning   |
|---------------------------|---|
| INRLOD/<br>NOINRL         | Controls calculation of residual vectors based on inertial forces due to rigid-body motion (Default =INRLOD).   |
| APPLOD/<br>NOAPPL         | Controls calculation of residual vectors based on applied loads (Default = APPLOD).   |
| ADJLOD/<br>NOADJLOD       | Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD).  |
| RVDOF/<br>NORVDOF         | Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF).   |
| DAMPLOD/<br>NODAMP        | Controls calculation of residual vectors based on viscous damping (Default = DAMPLOD).  |
| DYNRSP/<br>NODYNRSP       | Controls whether the residual vectors will be allowed to respond dynamically in the modal transient or frequency response solution. See Remark 5. (Default = DYNRSP).             |
| SYSTEM/<br>NOSYSTEM       | Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, descriptors inside the parentheses are ignored. See Remark 2. for default.                       |
| COMPONENT/<br>NOCOMPONENT | Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, descriptors inside the parentheses are ignored. See Remark 2. for default. |

| Descriptor  | Meaning  |
|-------------|--|
| BOTH or YES | Requests calculation of residual vectors for both system modes and component modes. See Remark 2. for default.   |
| NO          | Turns off calculation of residual vectors for both system and component modes, and describers inside the parentheses are ignored. See Remark 2. for default. |
| FLUID       | Requests residual vector calculation for the fluid model (Default = YES).  |

**Remarks:**

1. RESVEC=SYSTEM/NOSYSTEM and RESVEC=COMPONENT/NOCOMPONENT may be specified in the same subcase.
2. RESVEC=BOTH is the default in all solution sequences except SOLs 103, 106, (with PARAM,NMLOOP), and 115, wherein RESVEC=COMPONENT is the default.
3. If the RESVEC command is specified then the user parameters PARAM,RESVEC and PARAM,RESVINER are ignored.
4. The lower frequency cutoff on the EIGR or EIGRL Bulk Data entries should be left blank or set to a value below the minimum frequency. Residual vectors may not be calculated if all modes below the maximum frequency cutoff are not determined. If low frequency modes are to be excluded from the analysis, use the MODESELECT Case Control command or PARAM,LFREQ.
5. Caution needs to be exercised when allowing the residual vectors to respond dynamically in a modal solution. The best approach is to always include enough normal modes to capture the dynamics of the problem, and rely on the residual vectors to help account for the influence of the truncated modes on the quasistatic portion of the response. This is not the default setting for this capability. When choosing to allow the residual vectors to respond dynamically, it is important to be aware of the frequency content of the excitation, as it will have the ability to excite these augmentation modes. If this is undesirable, then the forcing function should be filtered in advance to remove any undesired frequency content, or specify the NODYNRSP keyword.
6. Residual vectors are normalized with respect to MASS if possible. Massless residual vectors are normalized with respect to STIFFNESS. Residual vectors are not normalized by the requested method of the NORM field of the selected EIGR or EIGRL entry.
7. The maximum number of applied loads that may be used for residual vector calculations is controlled by PARAM,MAXAPL. Note that if the number of applied loads is large, the cost can become prohibitive. If the number of applies loads exceeds MAXAPL, residual vectors are disabled.

**RGYRO**

Activates Gyroscopic Effects and Selects RGYRO or UNBALNC Entries

The RGYRO Case Control command activates the rotodynamics capability, and selects the RGYRO Bulk Data entry for use in complex modes, frequency response, and static analysis. For transient response, the RGYRO command selects the UNBALNC Bulk Data entry. If the UNBALNC entry is not required, setting RGYRO to YES will include the gyroscopic effects in the transient response calculation. Setting RGYRO to NO will deactivate gyroscopic effects in all solutions.

**Format:**

For complex modes, frequency response, and static analysis:

RGYRO = n or YES/NO

**Examples:**

RGYRO = 100

**For Transient Response:**

RGYRO = YES

or

RGYRO = 200

**RIGID****Rigid Element Method**

Selects rigid elements processing method for RBAR, RBAR1, RJOINT, RROD, RTRPLT, RTRPLT1, RBE1, RBE2, and RBE3.

**Format:**

$$RIGID = \begin{cases} LINEAR \\ LAGRAN \\ LGELIM \end{cases}$$

**Example:**

RIGID=LAGRAN

| Descriptor | Meaning  |
|------------|--|
| LINEAR     | Selects the linear elimination method.                   |
| LAGRAN     | Selects the Lagrange multiplier method.                  |
| LGELIM     | Selects the Lagrange multiplier method with elimination. |

**Remarks:**

1. The RIGID command must be above the SUBCASE level.
2. The RIGID command can be used in SOLs 101, 103, 105, and 400 only. For all other solution sequences, only RIGID=LINEAR is available.
3. If the RIGID command is not specified in the Case Control Section, RIGID=LINEAR is used for all solution sequences except SOL 400. For SOL 400, RIGID=LAGRAN is used. If the RIGID command is specified, the full command must be specified, including the right hand side.
4. RIGID=LGELIM is not available for SOL 400.
5. LINEAR processing will not compute thermal loads. Also, for SOLs 103 and 105, LINEAR processing will not compute differential stiffness. In order to compute thermal load or differential stiffness, the LAGRAN or LGELIM methods must be used.
6. For SOL 400, the LINEAR rigid elements are valid for small rotation only. The LAGRAN method is valid for both small and large rotation (parameter LGDISP=1).
7. For the LINEAR method, the dependent DOFs are eliminated and placed in the mp-set. For the LAGRAN method, both independent and dependent DOFs are placed in the l-set. Lagrange multiplier DOFs are created internally for the dependent DOFs and placed in l-set. For the LGELIM method, the LAGRAN rigid elements are created first. Then, both the Lagrange DOFs and the dependent DOFs are eliminated, and the dependent DOFs are placed in the mr-set. Both the mp-set and mr-set are subsets of the m-set. See [Degree-of-Freedom Set Definitions, 1112](#).
8. Between LAGRAN and LGELIM, LAGRAN is the preferred method. LGELIM is a backup method if difficulty is encountered using the LAGRAN method.

9. The parameters LMFAC and PENFN can be used as scale factor and penalty function, respectively, for the LAGRAN method of processing.
10. In a SOL 400 analysis with CWELD, CFAST, and/or CSEAM elements Nastran internally creates RBE3 elements. These internal RBE3 elements respond in the same way to the RIGID command as any other rigid body elements that may have been defined in the model. Therefore the RIGID command also has an effect on the behavior of CWELD and CFAST elements in a SOL 400 analysis.  
For "ANALYSIS=NSTAT" or "ANALYSIS=NTRAN", the generated RBE3 constraints become Lagrange elements and will undergo large rotation. For "ANALYSIS=NTRAN" with initial conditions (IC=n) in case control that cause large initial stresses in the structure at time t=0, the case control entry RIGID needs to have the value "RIGID=LINEAR." If "PARAM, OLDWELD, YES" is specified, the CSEAM is considered a linear element.
11. For external superelements, both the Case Control Command EXTSEOUT and the PARAM, EXTOUT, REQUIRE a value of RIGID=LINEAR (the default for non SOL400 solution sequences) when running non SOL 400 jobs.
12. When creating superelements in SOL 400, RIGID=LAGRAN produces incorrect answers, RIGID=LINEAR should be used or put the rigid elements into the residual.
13. When RIGID=LAGRAN is used, one may observe SWM 4968, indicating negative terms on multiple DOF which is due to the Lagrange multiplier. This message may be ignored and an accurate solution will be obtained.
14. When RIGID=LAGRAN is used, the CASI iterative solver may fail to converge or terminate, one should switch to a direct solver.

## RSDAMP

## Specifying Damping for the Residual Structure

Requests parameter and hybrid damping for the residual structure.

Format:

$$RSDAMP \left[ \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \\ \text{BOTH} \end{array} \right] = n$$

Example:

RSDAMP (STRUCTURE, FLUID, or BOTH) = n

| Descriptor | Meaning   |
|------------|---|
| n          | Identification number of a DAMPING Bulk Data entry (Integer > 0). |

Remarks:

1. For modal solutions, this entry adds to the modal damping that may be specified by the SDAMPING Case Control command.
2. This command can be different in each residual structure subcase.

**SACCELERATION****Solution Set Acceleration Output Request**

Requests the form and type of solution set acceleration output.

Format:

$$SACCELERATION \left[ \left( \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \text{PRINT}, \text{PUNCH}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

`SACCELERATION=ALL`

`SACCELERATION(PUNCH, IMAG)=142`

| Descriptor   | Meaning   |
|--------------|---|
| SORT1        | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.   |
| SORT2        | Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).  |
| PRINT        | The printer will be the output medium.  |
| PUNCH        | The punch file will be the output medium.   |
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.  |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| ALL          | Acceleration for all solution set points (modes) will be output.  |
| NONE         | Acceleration for no solution set points (modes) will be output.   |
| n            | Set identification number of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output (Integer > 0). |

Remarks:

1. Acceleration output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis, and are discussed in Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SACCELERATION=NONE allows an overall output request to be overridden.

**SDAMPING**

## Structural Damping Selection

Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.

Format:

$$SDAMPING \begin{bmatrix} STRUCTURE \\ FLUID \\ COUPLED \end{bmatrix} = n$$

Example:

SDAMPING=77  
SDAMPING (COUPLED)=100

| Descriptor            | Meaning  |
|-----------------------|--|
| STRUCTURE<br>or FLUID | Modal damping is requested for the structural or fluid portion of the model.         |
| COUPLED               | Modal damping is requested for the structure and fluid coupled portion of the model. |
| n                     | Set identification number of a TABDMP1 or TABLEDi Bulk Data entry (Integer>0).       |

Remarks:

1. In the modal solutions (e.g., SOLs 110, 111, 112, 145, 146, and 200), SDAMPING must reference a TABDMP1 entry.
2. In direct frequency response analysis (e.g., SOL 108), SDAMPING must reference a TABLEDi entry which defines viscoelastic (frequency-dependent) material properties. See [Viscoelastic Materials in Frequency Response Analysis](#) in the *MSC Nastran Reference Guide*.
3. When SDAMPING is defined for a superelement, PARAM,SESDAMP must be placed either in the SUBCASE for the superelement or in the superelement's BEGIN SUPER section in order to activate the modal damping for the superelement. The default for superelements is to place all boundary points in the B set; in this case, PARAM,SESDAMP,AUG is recommended. Alternatively, place all boundary points in the C set and set PARAM,SESDAMP,YES.
4. When SDAMPING is used during FRF component generation, PARAM,KDAMP,-1 is recommended in order to place the modal damping terms in the imaginary part of the stiffness matrix. The default for KDAMP is 0, which places the modal damping terms in the damping matrix, leading to a marginally under or over damped response. When hybrid damping is used during FRF component generation, it is recommended to set the KDAMP field to YES on the HYBDAMP entry.

5. When SDAMPING is used with METHOD(COUPLED), SDAMPING(COUPLED) should be used. For the real coupled mode computation, structural modes and fluid modes are not separated. So SDAMPING(STRUCTURE) or SDAMPING(FLUID) cannot be used properly. When SDAMPING(STRUCTURE/FLUID) is used with METHOD(COUPLED), it returns fatal out.

**SDISPLACEMENT****Solution Set Displacement Output Request**

Requests the form and type of solution set displacement output.

**Format:**

$$\text{SDISPLACEMENT} \left[ \begin{pmatrix} \text{SORT1} \\ \text{SORT2} \end{pmatrix}, \text{PRINT}, \text{PUNCH}, \begin{pmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{pmatrix} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

SDISPLACEMENT=ALL

SDISPLACEMENT (SORT2, PUNCH, PHASE) =NONE

| Descriptor   | Meaning   |
|--------------|---|
| SORT1        | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.   |
| SORT2        | Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).  |
| PRINT        | The printer will be the output medium.  |
| PUNCH        | The punch file will be the output medium.   |
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.  |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| ALL          | Displacements for all solution set points (modes) will be output.   |
| NONE         | Displacements for no solution set points (modes) will be output.  |
| n            | Set identification number of a previously appearing SET command. Only displacements on points with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. The defaults for SORT1 and SORT2 depend on the type of analysis, and is discussed in Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
2. SDISPLACEMENT=NONE allows an overall output request to be overridden.
3. The SDISPLACEMENT command is required to output normalized complex eigenvectors.

**SEALL**

## Superelement Generation and Assembly

Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.

Format:

$$SEALL = \left\{ \begin{array}{l} ALL \\ n \\ i \end{array} \right\}$$

Examples:

SEALL=ALL

SEALL=7

| Describer | Meaning  |
|-----------|--|
| ALL       | Generate and assemble all superelements.   |
| n         | Set identification number of a previously appearing SET command. Only superelements with identification numbers that appear on this SET command will be generated and assembled (Integer > 0). |
| i         | Identification number of a single superelement that will be generated and assembled (Integer > 0).   |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. This command combines, in one command, the functions of the SEMG, SELG, SEKR, SELR, and SEMR commands.
6. This command does not control superelement data recovery (Phase 3). See the [SEDR \(Case\)](#), 563 Case Control command description.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

## SEDAMP

### Specifying Damping for Superelements

Requests parameter and hybrid damping for superelements.

**Format:**

SEDAMP=n

**Example:**

| Descriptor | Meaning   |
|------------|---|
| n          | Identification number of a DAMPING Bulk Data entry (Integer > 0). |

**Remarks:**

1. This command adds to the modal damping that may be specified by the Case Control command SDAMPING, used in conjunction with PARAM,SESDAMP,YES.
2. Multiple SEDAMP requests per superelement are not allowed. If more than one SEDAMP request is specified per superelement, the second and subsequent requests will be ignored.

**SEDR****Superelement Data Recovery**

Specifies the superelement identification numbers for which data recovery will be performed.

Format:

$$SEDR = \left\{ \begin{array}{l} ALL \\ n \\ i \end{array} \right\}$$

Examples:

SEDR=ALL

SEDR=7

| Descriptor | Meaning   |
|------------|---|
| ALL        | Performs data recovery for all superelements.   |
| n          | Set identification number of a previously appearing SET command. Data recovery will be performed for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which data recovery will be performed (Integer > 0).   |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If this command is not present, data recovery is performed for all superelements for which there are output requests (i.e., the default for this command is SEDR=ALL).
6. The presence of the EXTDROUT Case Control command or the user PARAMeter EXTDROUT forces SEDR=ALL.

**SEDV****Superelement Design Variable Processing**

Specifies the superelement identification numbers for which the design variables will be processed.

**Format:**

$$SEDV = \left\{ \begin{array}{l} ALL \\ n \\ i \end{array} \right\}$$

**Examples:**

SEDV=ALL  
SEDV=18

| Descriptor | Meaning  |
|------------|--|
| ALL        | Requests design variable processing for all superelements. This is the default value if SEDV is missing in the file.   |
| n          | Set identification number of a previously appearing SET command. Design variable processing will be performed for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which design variable processing will be performed (Integer > 0).   |

**Remarks:**

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of superelement sensitivity analysis, see the *MSC.Nastran Design Sensitivity and Optimization User's Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

**SEEXCLUDE**

## Superelement Matrix and Load Assembly Exclusion

Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.

Format:

$$\text{SEEXCLUDE} = \begin{Bmatrix} \text{ALL} \\ n \\ i \end{Bmatrix}$$

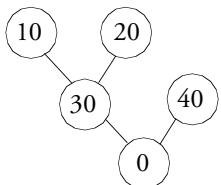
Examples:

`SEEXCLUDE=ALL`  
`SEEXCLUDE=18`

| Describer | Meaning   |
|-----------|---|
| ALL       | All upstream superelements will be excluded.  |
| n         | Set identification number of a previously appearing SET command. Only those superelements with identification numbers that appear on this SET command will be excluded (Integer > 0). |
| i         | Identification number of a single superelement for which matrices will be excluded (Integer > 0).   |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
3. This command is not meaningful when applied to the residual structure.
4. For a further discussion of this command, see the *MSC Nastran Handbook for Superelement Analysis*.
5. If the SEEXCLUDE command is specified in a restart of SOLs 101 through 200, then PARAM,SERST,MANUAL must be specified. Also, the SEKR command must be specified for the superelement immediately downstream from the excluded superelement. For example, if superelement 10 is excluded in the following superelement tree:



then the user must specify the following commands in the Case Control Section:

`SEKR = 30`  
`PARAM, SERST, MANUAL`

**SEFINAL**

## Final Superelement for Assembly

Specifies the superelement identification number for the final superelement to be assembled.

Format:

$$\text{SEFINAL} = \left\{ \begin{array}{c} n \\ i \end{array} \right\}$$

Example:

SEFINAL=14

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a previously appearing SET command. Each superelement identification number appearing on the SET command must belong to a disjoint model (Integer > 0). |
| i          | Identification number of the final superelement to be assembled (Integer > 0).  |

Remarks:

1. If this command is not present, the program selects the order of the superelements for assembly operations.
2. This command, if present, must be located before the first SUBCASE command.
3. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
4. This command can be used on restarts to ensure that minor modeling changes do not also change the processing order. For this usage, inspect the SEMAP table to determine which superelements were final superelements on the prior run.
5. See the *MSC Nastran Handbook for Superelement Analysis* for a further discussion of this command.

**SEKREDUCE**

## Superelement Stiffness Matrix Assembly and Reduction

Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.

Format:

$$\text{SEKREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEKREDUCE=ALL

SEKREDUCE=9

| Descriptor | Meaning   |
|------------|---|
| ALL        | Assembles and reduces matrices for all superelements.   |
| n          | Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which the stiffness matrix will be assembled and reduced (Integer > 0).  |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. SEKREDUCE is an alternate form, and is entirely equivalent to the obsolete command SEMASSEMBLE.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SELGENERATE**

## Superelement Load Generation

Specifies the superelement identification numbers for which static loads will be generated.

Format:

$$\text{SELGENERATE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

`SELGENERATE=ALL`  
`SELGENERATE=18`

| Descriptor | Meaning   |
|------------|---|
| ALL        | Generates static loads for all superelements.   |
| n          | Set identification number of a previously appearing SET command. Static load matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which load matrices will be generated (Integer > 0).   |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SELREDUCE****Superelement Load Assembly and Reduction**

Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.

Format:

$$\text{SELREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

`SELREDUCE=ALL`  
`SELREDUCE=9`

| Describer | Meaning   |
|-----------|---|
| ALL       | Assembles and reduces matrices for all superelements.   |
| n         | Set identification number of a previously appearing SET command. Matrices will be assembled only for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i         | Identification number of a single superelement for which the load matrices will be assembled and reduced (Integer > 0).   |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only be appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. This command is used on restarts to selectively assemble and reduce load matrices.
5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
6. In superelement static analysis, SELREDUCE is equivalent to SELASSEMBLE.
7. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
8. SEALL=ALL is the default, but can be overridden by specifying and of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SEMGENERATE****Superelement Matrix Generation**

Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.

Format:

$$\text{SEMGENERATE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEMGENERATE=ALL  
SEMGENERATE=7

| Descriptor | Meaning  |
|------------|--|
| ALL        | Generates structural matrices for all superelements.   |
| n          | Set identification number of a previously appearing SET command. Structural matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which structural matrices will be generated (Integer > 0).  |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, in the structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SEMREDUCE****Superelement Mass and Damping Assembly and Reduction**

Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices will be assembled and reduced.

Format:

$$\text{SEMREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEMREDUCE=ALL

SEMREDUCE=9

| Describer | Meaning   |
|-----------|---|
| ALL       | Assembles and reduces matrices for all superelements.   |
| n         | Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i         | Identification number of a single superelement for which the load matrices or the mass and damping matrices will be assembled and reduced (Integer > 0).                                      |

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a set.
3. This command is used on restart to selectively assemble and reduce mass and damping matrices. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
6. This command has no function in static analysis.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SERESP****Superelement Response Sensitivity**

Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.

**Format:**

$$SERESP = \left\{ \begin{array}{l} ALL \\ n \\ i \end{array} \right\}$$

**Examples:**

SERESP=ALL  
SERESP=18

| Descriptor | Meaning   |
|------------|---|
| ALL        | Requests design sensitivity matrix generation for all superelements. This is the default value if SERESP is missing.  |
| n          | Set identification number of a previously appearing SET command. Design sensitivity matrices will be generated for superelements with identification numbers that appear on this SET command (Integer > 0). |
| i          | Identification number of a single superelement for which the design sensitivity matrix will be generated.   |

**Remarks:**

1. This command, if present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see the [MSC Nastran Reference Manual](#).
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

**SET**

## Set Definition, General Form

Sets are used to define the following lists:

1. Identification numbers (point, element, or superelement) for processing and output requests.
2. Frequencies for which output will be printed in frequency response problems, or times for transient response, using the OFREQ and OTIME commands, respectively.
3. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD commands.
4. DRESP1 entries that are used in the spanning of subcases.
5. Grid point number and component type code to be used by the MCFRACTION command.

**Formats:**

*SET n* = {*i*<sub>1</sub>[,*i*<sub>2</sub>, *i*<sub>3</sub>, THRU *i*<sub>4</sub>, EXCEPT *i*<sub>5</sub>, *i*<sub>6</sub>, *i*<sub>7</sub>, *i*<sub>8</sub>, THRU *i*<sub>9</sub>]}

*SET n* = {*r*<sub>1</sub>[,*r*<sub>2</sub>, *r*<sub>3</sub>, *r*<sub>4</sub>]}

*SET* = ALL

*SET n* = {*i*<sub>1</sub>/*c*<sub>1</sub>[,*i*<sub>2</sub>/*c*<sub>2</sub>, *i*<sub>3</sub>/*c*<sub>3</sub>, *i*<sub>4</sub>/*c*<sub>4</sub>]}

*SET n* = {*l*<sub>1</sub>, [*l*<sub>2</sub>, *l*<sub>3</sub>]}

**Examples:**

```
SET 77=5
SET 88=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU
300
SET 99=1 THRU 100000
```

```
SET 101=1.0, 2.0, 3.0
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
SET 1001=101/T1, 501/T3, 991/R3
SET 2001=M1,M2
```

| Descriptor   | Meaning  |
|--|--|
| n  | Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0). |
| <i>i</i> <sub>1</sub> , <i>c</i> <sub>1</sub>      | Grid point identification numbers and component codes. The <i>c</i> values must be of T1, T2, T3, R1, or R3.   |
| <i>l</i> <sub>1</sub> , <i>l</i> <sub>2</sub> etc. | Identification names of literals used for matrix or group selection.   |
| <i>i</i> <sub>1</sub> , <i>i</i> <sub>2</sub> etc. | Identification numbers. If no such identification number exists, the request is ignored (Integer $\geq$ 0).  |
| <i>i</i> <sub>3</sub> THRU <i>i</i> <sub>4</sub>   | Identification numbers ( <i>i</i> <sub>4</sub> > <i>i</i> <sub>3</sub> ) (Integer > 0).  |

| Descriptor              | Meaning  |
|-------------------------|--|
| EXCEPT                  | Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.   |
| $r_1, r_2, \text{etc.}$ | Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ or OTIME command references the set then the values must be listed in ascending sequences, $r_1 < r_2 < r_3 < r_4 \dots \text{etc.}$ , otherwise some output may be missing. If an OFREQ or OTIME command is not present, all frequencies or times will be output (Real $> 0.0$ ). |
| ALL                     | All members of the set will be processed.  |

**Remarks:**

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.
2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.
3. In SET 88 above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.
4. SET commands using the grid point/component code format cannot contain THRU. SETs using this format should be selected only by the MCFRACTION Case Control command.
5. SET commands using literals apply only to direct matrix input such as K2PP etc. or FLSPROUT panel grouping.

**SETP****Process Set Definition**

Process sets are used to define lists of SET identifications to be processed individually for data recovery:

**Formats:**

*SETP n = {i<sub>1</sub>[i<sub>2</sub>, i<sub>3</sub>THRU i<sub>4</sub>EXCEPT i<sub>5</sub>, i<sub>6</sub>, i<sub>7</sub>, i<sub>8</sub>THRU i<sub>9</sub>]}*

**Examples:**

SETP 77=5, 6

SETP 88=5, 6, 7, 8, 9, 10 THRU 55

| Descriptor  | Meaning  |
|---|--|
| n   | SETP identification number. Any SETP may be redefined by reassigning its identification number. SETPs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0).                            |
| i <sub>1</sub> , i <sub>2</sub> , ..., i <sub>n</sub> | SET identification numbers i <sub>1</sub> , i <sub>2</sub> , etc. If no such identification number exists, the request is ignored (Integer > 0).   |
| EXCEPT  | Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL. |

**Remarks:**

1. A SETP command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation—place a number after the THRU.
2. Set identification numbers following EXCEPT,k within the range of the THRU, must be in ascending order.
3. SETP usage is limited to the EDE, EKE and ESE Case Control commands.

## SETS DEFINITION

## Case Control Processing Delimiter

Delimits the various type of commands under grid point stress and/or p-version element set definitions. This command is synonymous with OUTPUT(POST).

**Format:**

SETS DEFINITION

**Example:**

SETS DEFINITION

**Remark:**

1. Either SETS DEFINTIIION or OUTPUT(POST) may be specified, but not both.

**SKIP**

## Case Control Processing Delimiter

Activates or deactivates the execution of subsequent commands in Case Control (including plot commands).

Format:

*SKIP* { *ON* }  
          { *OFF* }

Example:

SKIPOFF

Remarks:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control Section.
2. Commands that are skipped will be printed.
3. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows the user to omit requests without deleting them from the data. In the following example, plot commands will be skipped.

```
TITLE=EXAMPLE
SPC=5
LOAD=6
SKIPON$SKIP PLOT REQUEST
OUTPUT (PLOT)
SET 1 INCLUDE ALL
FIND
PLOT
BEGIN BULK
```

**SMETHOD**

## Iterative Solver Method Selection

Selects iterative solver method and parameters.

Format:

$$\text{SMETHOD} \left\{ \begin{array}{l} \text{ELEMENT} \\ n \\ \text{MATRIX} \end{array} \right\}$$

Example:

```
SMETHOD = ELEMENT $ selects element-based iterative solver defaults.
SMETHOD = MATRIX $ selects matrix based iterative solver defaults.
SMETHOD = 1000 $ specifies ID of ITER Bulk Data entry to select
iterative.
```

| Descriptor | Meaning   |
|------------|---|
| ELEMENT    | Selects the element-based iterative solver with default control values. |
| MATRIX     | Selects the matrix-based iterative solver with default control values.  |
| n          | Sets identification of an ITER Bulk Data entry (Integer > 0).           |

Remarks:

1. The matrix-based iterative solver is available in SOLs 101, 106, 108, 111, 153, and 400 and allows use of all features.
2. The element-based iterative solver is only available in SOLs 101, 200 and 400. SMETHOD must be placed above all SUBCASEs in this case. It is intended primarily for very large solid element models and does not handle p-elements. See the ITER Bulk Data entry for a list of restrictions in addition to details on setting the convergence parameter epsilon.
3. The element-based iterative solver can be used with SMP by setting smp=number on the command line. Please refer to [smp, 11](#) of this manual for correct usage.
4. For SOL 600, the iterative solver is activated using the MARCSOLV PARAM.
5. GPGPU devices are not supported for iterative methods.

**SOLUTION****Solution ID Selection**

Selects the solution ID for a 3<sup>rd</sup> step external superelement data recovery restart in SOL 400.

**Format:**

**SOLUTION = n**

**Example:**

**SOLUTION=10**

| Descriptor | Meaning                         |
|------------|---------------------------------|
| n          | Solution identification number. |

**Remarks:**

1. SOLUTION command is used only in SOL 400 for 3<sup>rd</sup> step external superelement data recovery restart.
2. See Remark 8. under EXTDRIN Case Control command's description for a further explanation and example.

## SPC Single Point Constraint Set Selection

Selects a single point constraint set to be applied.

**Format:**

SPC = n

**Example:**

SPC=10

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of a single-point constraint that appears on an SPC, SPC1, SPC2 (SOL 700), FRFSPC1 (in FRF Based Assembly or FBA process) or SPCADD Bulk Data entry (Integer > 0). |

**Remarks:**

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary conditions are only supported in SOLs 101, 103, 105, 145, and 200. Multiple boundary conditions are not allowed for upstream superelements. The BC command must be specified to define multiple boundary conditions for the residual structure in SOLs 103, 105, 145, and 200.

**SPCFORCES**

## Single Point Forces of Constraint Output Request

Requests the form and type of single point force of constraint vector output.

Format:

$$\text{SPCFORCES} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or NOZPRINT} \\ \text{PHASE} \end{array} \right] \left[ \begin{array}{c} \text{PSDF}, \text{ATOC}, \text{CRMS} \\ \text{or RALL} \end{array} \right], \\ \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{CID} \end{array} \right] \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

SPCFORCES = 5  
 SPCFORCES(SORT2, PUNCH, PRINT, IMAG) = ALL  
 SPCFORCES(PHASE) = NONE  
 SPCFORCES(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20  
 SPCFORCES(PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |

| Descriptor | Meaning  |
|------------|--|
| NOZPRINT   | Print only nonzero SPC forces appearing in SORT2 output. This keyword does not affect SORT1 output.  |
| PSDF       | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9. |
| ATOC       | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.            |
| CRMS       | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.                           |
| RALL       | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.  |
| RPRINT     | Writes random analysis results in the print file (Default).  |
| NORPRINT   | Disables the writing of random analysis results in the print file.   |
| PRUNCH     | Writes random analysis results in the punch file.  |
| CID        | Request to print output coordinate system ID in printed output file (.f06).  |
| ALL        | Single point forces of constraint for all points will be output. See Remarks 1. and 4.   |
| NONE       | Single point forces of constraint for no points will be output.  |
| n          | Set identification of a previously appearing SET command. Only single point constraint forces for points with identification numbers that appear on this SET command will be output (Integer > 0).                                     |

### Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
3. SPCFORCES=NONE overrides an overall output request.
4. In SORT1 format, SPCFORCES recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
5. SPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
6. In SOLs 129 and 159, SPCFORCES results do not include the effects of mass and damping elements.

7. In all solution sequences except SOLs 129 and 159, SPCFORCES results do include the effects of mass and damping, except damping selected by the SDAMPING Case Control command. PARAM,DYNNSPCF,OLD may be specified to obtain SPCFORCES results, which do not include mass and damping effects.
8. In inertia relief analysis, the SPCFORCES output is interpreted differently for SOLs 1, 101, and 200:
  - a. In SOL 1, the SPCFORCE output reflects the effects due to the applied loads only, and not the inertial loads.
  - b. In SOLs 101 and 200, the SPCFORCE output includes both the effects due to inertial loads and applied loads.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Note that the CID keyword affects only grid point related output such as DISP, VELO, ACCE, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid-related output command, anywhere in the Case Control Section, to turn on the printing algorithm.

**SPLINOUT**

## Export of Spline Matrix

Requests output of the spline matrix for external use.

**Format:**

$$\text{SPLINOUT} \left[ \left( \begin{array}{c} \text{BOTH} \\ \text{DISP} \\ \text{FORCE} \end{array} \right), \left\{ \begin{array}{c} (\text{OP2} = \text{unit}) \\ \text{DMIPCH} \end{array} \right\} \right]$$

**Examples:**

| Descriptor | Meaning  |
|------------|--|
| BOTH       | Requests output of the force and displacement splines (Default). |
| DISP       | Requests output of only the displacement splines.                |
| FORCE      | Requests output of only the force splines.                       |
| OP2        | Requests output to an .op2 file                                  |
| unit       | Unit the .op2 file is assigned to                                |
| DMIPCH     | Requests output to a .pch file.                                  |

**Remarks:**

1. Matrices are output in external sort.
2. If displacement and force splines are identical, only displacement splines are output.
3. Option FACTORS is only allowed together with option OP2.
4. If OP2 = *unit* is specified, a table relating the matrix columns to structural degrees of freedom, and the matrix rows to aerodynamic degrees of freedom, will be written to the .op2 file.
5. If OP2 = *unit* is specified, an appropriate ASSIGN OP2 statement must be present in the File Management Section for this unit.
6. If DMIPCH is specified, DMI entries are written to the.pch file.

**STATSUB**

## Static Solution Selection for Differential Stiffness

Selects the static solution to use in forming the differential stiffness for static analysis, buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.

Format:

$$\text{STATSUB} \left( \begin{matrix} \text{BUCKLING} \\ \text{PRELOAD} \end{matrix} \right) = n$$

Examples:

`STATSUB=23`

`STAT=4`

`STATSUB (PREL)=7`

| Descriptor | Meaning   |
|------------|---|
| BUCKLING   | Subcase ID number corresponding to static subcase of buckling or varying load (Default in buckling analysis). |
| PRELOAD    | Subcase ID number corresponding to static subcase of preload or constant load (Default in dynamic analysis).  |
| n          | Subcase identification number of a prior subcase specified for static analysis (Integer > 0).                 |

Remarks:

1. STATSUB may be used in SOLs 101, 103, 105, 107 through 112, 115, 116, 200 and SOL 400 (ANALYSIS = BUCKLING only in SOL 200 and SOL 400).
2. STATSUB must be specified in the same subcase that contains the METHOD selection for buckling or normal modes, CMETHOD for complex eigenvalue analysis, TSTEP for transient response, and FREQ for frequency response.
3. In SOL 105, if it is intended that results from the first static subcase are used to compute the differential stiffness, then the STATSUB command is not required. That is, the default for STATSUB is the first static subcase identification. In SOLs 101, 103 and 107 through 112, 115, and 116, STATSUB must reference a separate static subcase.
4. In dynamic analysis, only one STATSUB command may be specified in each dynamic subcase. In buckling analysis with a preload, both STATSUB(BUCKLING) and STATSUB(PRELOAD) must be specified in each buckling subcase. STATSUB(PRELOAD) is not supported in SOL 200 or SOL 400. Buckling Analysis with a preload is not supported in SOL 200 and SOL 400.
5. In dynamic analysis, any subcase that does not contain a CMETHOD command in SOLs 107 and 110, a FREQUENCY command in SOLs 108 and 111, and a TSTEP command in SOLs 109 and 112, will be treated as a static subcase.
6. SOL 200 and SOL 400 support linear buckling analysis only, but do not support post-buckling (nonlinear buckling analysis)

7. In versions prior to 2018, if it was desired to have a dynamic subcase **with** the effects of preload and another subcase **without** the effects of preload, then a static subcase with null load was required and its ID was referenced by STATSUB in the dynamic subcase **without** the effects of preload. In version 2018, STATSUB in the dynamic subcase **without** the effects of preload is no longer required. If the STATSUB is not removed and still references the static subcase with null load, then User Fatal Message 9244 will be issued.

**STEP****Step Delimiter**

Delimits and identifies a nonlinear analysis step for SOL 400.

**Format:**

STEP=n

**Examples:**

STEP=10

| Descriptor | Meaning                                   |
|------------|---|
| n          | Step identification number (Integer > 0). |

**Remarks:**

1. The STEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
2. The STEP command is to be used below the SUBCASE Case Control command. If no SUBCASE is specified, MSC Nastran creates a default SUBCASE 1.
3. The STEP identification number n in a SUBCASE must be in increasing order, and less than 9999999.
4. The following example illustrates a typical application of SUBCASE and STEP:

```
SUBCASE 1
    STEP 1
        LOAD = 1
    STEP 2
        LOAD = 2
SUBCASE 2
    STEP 10
        LOAD = 10
    STEP 20
        LOAD = 20
```

5. The solutions of all SUBCASEs are independent of each other. However, the solution of any STEP is a continuation of the solution of the previous STEP.

## STOCHASTICS

### Randomization of Model Parameters

Request randomization of all or selected subsets of model parameters.

Format:

$$STOCHASTICS = \begin{Bmatrix} ALL \\ n \end{Bmatrix}$$

Examples:

STOCHASTICS=10

| Descriptor | Meaning   |
|------------|---|
| ALL        | All real values of C-entries, M-entries, P-entries, loading entries, and SPCD entries are to be randomized. |
| n          | Set identification number of a STOCHAS Bulk Data entry (Integer > 0).                                       |

Remarks:

1. Only one STOCHASTICS command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. The STOCHASTICS = n command may be used to request randomizing a set of analysis model parameters with user specified statistics. (See Remark 1 of the STOCHAS Bulk Data entry.)
3. The default (STOCHASTICS = all) randomizes all scalar analysis model parameters that are real values on the C-entries, M-Entries, P-entries, all loading entries, and SPCD entries with default coefficients of variance (0.05) and multipliers of standard deviations (m=3.).
4. This command will only invoke a single Nastran randomization run. Separate runs can be submitted to achieve different randomizations.

**STRAIN**

## Element Strain Output Request

Requests the form and type of strain output.

Format:

$$\text{STRAIN} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{c} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \left[ \begin{array}{c} \text{STRCUR} \\ \text{FIBER} \end{array} \right], \right. \right. \\ \left. \left. \begin{array}{c} \left[ \begin{array}{c} \text{CENTER} \\ \text{CORNER or BILIN} \\ \text{SGAGE} \\ \text{CUBIC} \end{array} \right], \left[ \begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{RPUNCH} \end{array} \right] \end{array} \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\} \right]$$

Examples:

STRAIN=5  
 STRAIN(CORNER)=ALL  
 STRAIN(PRINT, PHASE)=15  
 STRAIN(PLOT)=ALL  
 STRAIN(PRINT, PSDF, CRMS, RPUNCH)=20  
 STRAIN(PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each element.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |

| Descriptor      | Meaning  |
|-----------------|--|
| PSDF            | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9. |
| ATOC            | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.        |
| CRMS            | Requests the cumulative root mean square function to be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.                        |
| RALL            | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.  |
| VONMISES        | von Mises strains are output.  |
| MAXS or SHEAR   | Maximum shear strains are output.  |
| STRCUR          | Strain at the reference plane and curvatures are output for plate elements.  |
| FIBER           | Strain at locations Z1, Z2 are computed for plate elements.  |
| CENTER          | Output CQUAD4 element strains at the center only.  |
| CORNER or BILIN | Output CQUAD4 element strains at the center and grid points. Using strain gage approach with bilinear extrapolation.   |
| SGAGE           | Output CQUAD4 element strains at center and grid points using strain gage approach.  |
| CUBIC           | Output CQUAD4 element strains at center and grid points using cubic bending correction.  |
| RPRINT          | Writes random analysis results in the print file (Default).  |
| NORPRINT        | Disables the writing of random analysis results in the print file.   |
| RPUNCH          | Writes random analysis results in the punch file.  |
| ALL             | Strain for all elements will be output.  |
| n               | Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output (Integer > 0).   |
| NONE            | No element strain will be output.  |

**Remarks:**

1. Fully accurate nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output. In SOLs 106 and 129, if LGDISP > 0 the center, and if requested for CQUAD4 the corner, strains are computed from the displacements alone and are only approximate. If LGDISP = -1 and material nonlinear then no STRAIN output will be created. In SOL 400, if enhanced material options are used, the computed strains are accurate and in this case a CQUAD4 corner request is ignored.
2. In SOLs 106 and 129, the STRAIN request pertains only to linear elements and only if the parameter LGDISP is -1, which is the default. Nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output.
3. STRAIN=NONE overrides an overall output request.
4. Definitions of stress, strain, curvature, and output locations are given in the [Structural Elements](#) in the *MSC Nastran Reference Guide*.
5. If the STRCUR option is selected, the values of Z1 will be set to 0.0. and Z2 will be set to -1.0 on the output.
6. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\), 382](#) Case Control command for further discussion.
8. See Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command for a discussion of SORT1 and SORT2.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Random response for complex strain in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.

**STRESS****Element Stress Output Request**

Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.

**Format:**

$$\text{STRESS} \left[ \begin{array}{c} \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{c} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \right. \\ \left. \begin{array}{c} \left[ \begin{array}{c} \text{CENTER} \\ \text{CUBIC} \\ \text{SGAGE} \\ \text{CORNER or BILIN} \end{array} \right], \left[ \begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[ \begin{array}{c} \text{RPUNCH} \\ \text{RPUNCH} \end{array} \right] \end{array} \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

```
STRESS=5
STRESS (CORNER)=ALL
STRESS (SORT1,PRINT,PUNCH,PHASE)=15
STRESS (PLOT)=ALL
STRESS (PRINT, PSDF, CRMS, RPUNCH)=20
STRESS (PRINT, RALL, NORPRINT)=ALL
```

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each element type.  |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG

Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

| Descriptor         | Meaning   |
|--------------------|---|
| PHASE              | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| PSDF               | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11. |
| ATOC               | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.        |
| CRMS               | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.                       |
| RALL               | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.                                    |
| VONMISES           | Requests von Mises stresses.  |
| MAXS or<br>SHEAR   | Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.  |
| CENTER             | Requests CQUAD4, CQUADR and CTRIAR element stresses at the center only. The default for CQUAD4 is CENTER. The default for CQUADR and CTRIAR is CORNER.  |
| CUBIC              | Requests CQUAD4 element stresses at the center and grid points using strain gage approach with cubic bending correction.  |
| SGAGE              | Requests CQUAD4 element stresses at center and grid points using strain gage approach.  |
| CORNER or<br>BILIN | Requests CQUAD4, CQUADR. and CTRIAR element stresses at center and grid points using bilinear extrapolation.  |
| RPRINT             | Writes random analysis results in the print file (Default).   |
| NORPRINT           | Disables the writing of random analysis results in the print file.  |
| RPUNCH             | Writes random analysis results in the punch file.   |
| ALL                | Stresses for all elements will be output.   |
| n                  | Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output (Integer > 0).  |
| NONE               | No element stress will be output.   |

**Remarks:**

1. ALL should not be used in a transient problem due to excessive output.

2. See Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command description for a discussion of SORT1 and SORT2.
3. ELSTRESS is an alternate form and is equivalent to STRESS.
4. STRESS=NONE overrides an overall output request.
5. Prior to MSC Nastran 2014, in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.
6. The VONMISES option is ignored for ply stresses.
7. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\), 382](#) Case Control command for further discussion.
9. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).
10. MAXS for shell elements is not an equivalent stress.
11. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
12. Random response for complex stress in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.

**STRFIELD****Grid Point Stress Output Request**

Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.

Format:

$$\text{STRFIELD} = \left\{ \begin{array}{l} \text{ALL} \\ n \end{array} \right\}$$

Examples:

STRFIELD=ALL  
STRFIELD=21

| Descriptor | Meaning   |
|------------|---|
| ALL        | Grid point stress requests for all surfaces and volumes defined in the OUTPUT(POST) Section will be saved for postprocessing.   |
| n          | Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command, and in the OUTPUT(POST) Section, will be included in the grid point stress output request for postprocessing (Integer > 0). |

Remarks:

1. The STRFIELD command is required for the graphical display of grid point stresses in postprocessors that use the .xdb file (PARAM,POST,0), or when the GPSDCON or ELSDCON commands are specified, and does not provide printed output. The GPSTRESS command can be used to obtain printed output.
2. Only grid points connected to elements used to define the surface or volume are output. See the SURFACE and VOLUME Case Control commands.
3. Element stress output (STRESS) must be requested for elements referenced on requested SURFACE and VOLUME Case Control commands.
4. In nonlinear static and transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default. Also, in nonlinear transient analysis, grid point stresses are computed only for elements with linear material properties.

## SUBCASE

### Subcase Delimiter

Delimits and identifies a subcase.

**Format:**

SUBCASE=n

**Example:**

SUBCASE=101

| Descriptor | Meaning  |
|------------|--|
| n          | Subcase identification number (9999999 > Integer > 0). |

**Remarks:**

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot requests and RANDPS requests refer to n.
3. See the MODES Case Control command for use of this command in normal modes analysis.
4. If a comment follows n, then the first few characters of the comment will appear in the subcase label in the upper right-hand corner of the output.
5. Note that in nonlinear statics (SOL106/129), SUBCASE's are not stand-alone solutions like in other solution sequences. They act as a load-progression, and the ending conditions of one SUBCASE become the initial conditions of the next SUBCASE.

**SUBCOM****Combination Subcase Delimiter**

Delimits and identifies a combination subcase.

**Format:**

SUBCOM = n

**Example:**

SUBCOM = 125

| Descriptor | Meaning                                      |
|------------|--|
| n          | Subcase identification number (Integer > 2). |

**Remarks:**

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. A SUBSEQ command must follow this command.
3. SUBCOM may only be used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
4. Output requests above the subcase level will be used.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SUBCOM with a TEMP(LOAD) command, or the element deformations with a DEFORM command.
6. SUBCOMs may be specified in superelement analysis with the following recommendations:
  - a. For each superelement, specify its SUBCASE(s) consecutively, directly followed by its SUBCOM(s).
  - b. Specify a SUPER command with a new load sequence number under each SUBCOM command.

The following example demonstrates a model with one superelement and one load combination:

```

SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SUBCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SUBSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SUBCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SUBSEQ=1.,1.
  
```

7. For static aeroelasticity, only the displacement/element responses are combined. Trim, stability derivative and monitor point results are not combined.
8. SUBCOMs are not allowed in external superelement creation runs and are ignored in external superelement data recovery in assembly runs.

**SUBSEQ****Subcase Sequence Coefficients**

Gives the coefficients for forming a linear combination of the previous subcases.

**Format:**

SUBSEQ=R1 [, R2, R3, ..., Rn]

**Example:**

SUBSEQ=1.0, -1 .0, 0.0, 2.0

| Descriptor | Meaning  |
|------------|--|
| Ri         | Coefficients of the previously occurring subcases. See Remark 4. (Real). |

**Remarks:**

1. The SUBSEQ command can only appear after a SUBCOM command.
2. SUBSEQ may be only used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
3. This command list is limited to a maximum of 200 numbers.
4. R1 to Rn refer to the immediately preceding subcases. In other words, Rn is applied to the most recently appearing subcase, R(n - 1) is applied to the second most recently appearing subcase, and so on. The embedded comments (\$) describe the following example:

```
DISPL = ALL
SUBCASE 1
SUBCASE 2
SUBCOM 3
SUBSEQ = 1.0, -1.0 $ SUBCASE 1 - SUBCASE 2
SUBCASE 11
SUBCASE 12
SUBCOM 13
SUBSEQ = 0.0, 0.0, 1.0, -1 .0 $ SUBCASE 11 - SUBCASE 12
or
SUBSEQ = 1.0, - 1.0 $ EQUIVALENT TO PRECEDING COMMAND. USE ONLY
ONE.
```

## SUBSEQ1

### Subcase Factors for Combination

Gives the factors for linear combination of a specific group of SUBCASEs.

**Format:**

SUBSEQ1= s0, s1, sub1, s2, sub2, [sn, subn]

**Example:**

SUBSEQ1= 1.0, 1.0, 101, -1.0, 102

| Descriptor | Meaning   |
|------------|---|
| S0         | Factor for all SUBCASEs involved. (No default; Real<>0.). |
| Sn         | Factor applicable to SUBn only (Real; No default).        |
| SUBn       | SUBCASE ID (No default; Integer>0).                       |

**Remarks:**

1. The SUBSEQ1 command can only appear after a SUBCOM command.
2. SUBSEQ1 may only be used in SOL 101 (Statics) or SOL 144 (Static Aeroelasticity) and in SOL 200 with ANALYSIS=STATIC or ANALYSIS=SAERO.
3. SUBSEQ1 and SUBSEQ are mutually exclusive and can't both appear under a SUBCOM.
4. S0, S1 and SUB1are required input for SUBSEQ1. S2,SUB2 to Sn,SUBn pair are optional.

**SUBSTEP****SUBSTEP Delimiter**

Delimits and identifies a nonlinear analysis SUBSTEP for COUPLED analysis in SOL 400.

**Format:**

SUBSTEP=n

**Examples:**

STEP=50

NLSTEP=10

SUBSTEP=1

ANALYSIS=HSTAT

SUBSTEP=2

ANALYSIS=NLSSTAT

| Descriptor | Meaning                                      |
|------------|--|
| n          | Substep identification number. (Integer > 0) |

**Remarks:**

1. The SUBSTEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
  2. The SUBSTEP command can only be used in a STEP command.
  3. When used in a STEP command two or more SUBSTEP commands must occur.
  4. Each SUBSTEP must contain a *unique* ANALYSIS-type statement. Currently there is a limitation of only two SUBSTEPS per STEP with the following options:
    - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLSSTAT for the second SUBSTEP.
    - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLTTRAN for the second SUBSTEP
    - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLSSTAT for the second SUBSTEP
    - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLTTRAN for the second SUBSTEP
  5. Within a STEP the SUBSTEP identification number n must be in increasing order and not greater than 9999999.
  6. The following example illustrates a typical application of SUBCASE, STEP, and SUBSTEPS
- Subcase 100

```
STEP 10
STRESS= ALL
NLSTRESS=ALL
SET 1456 = list
NLSTEP=84
SubSTEP 1
```

```

ANALYSIS=HSTAT
THERMAL=ALL
FLUX=ALL
SPC=35
LOAD=11
SubSTEP 2
ANALYSIS=NLSTAT
SPC=2
LOAD=110
DISP(PLOT)=1456

```

7. The solutions of all SUBCASEs are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP. *The solutions of the SUBSTEPS occur simultaneously within a STEP.*
8. In coupled analysis, TEMP(LOAD)=m will be ignored as the temperature loading from the thermal substep is automatically transferred to the mechanical substep. Note that in this multi-physics coupled framework, both the heat transfer and structural physics are executed on the same mesh with the same time steps.
9. In coupled analysis, TEMP(INIT)=n will be honored. It should be noted that when a thermo-mechanical static analysis is defined, TEMP(INIT) defines the initial temperature for both the thermal and mechanical substeps. Note that when a transient thermal-static mechanical analysis is define, IC=n should be used to define the initial temperature for the thermal substep and TEMP(INIT)=n (the same ID n) should be used to define the initial temperature for the structural substep. Then an initial thermal strain is defined as:

$$\varepsilon T = A(T) \cdot (T - TREF) - A(T_o) \cdot (T_o - TREF)$$

where  $T_o$  comes from TEMP(INIT) and TREF comes from the material entry.

10. The LOADSET Case Control Command *is not allowed* with this command. All dynamic loading must be applied through the use of the DLOAD Case Control command.
11. For coupled analysis, i.e., when two or more sub-steps are defined, a *single* Bulk Data entry NLSTEP must be included in the STEP command containing the SUBSTEP commands. It must occur above or in the STEP command and above the first SUBSTEP command.

The loads and constraints can be independently defined for each physics under the relevant SUBSTEP. The commands that are normally used for single physics are applicable. For, e.g., for a HTRAN-NLSTAT coupled analysis, DLOAD should be used n the HTRAN substep while LOAD should be used in the NLSTAT substep.

12. Any case control command will apply to all SUBSTEPS when and only when it is above all the SUBSTEPS of the STEP.

**SUBTITLE**

Output Subtitle

Defines a subtitle that will appear on the second heading line of each page of printer output.

**Format:**

SUBTITLE=subtitle

**Example:**

SUBTITLE=PROBLEM NO. 5-1A

| Descriptor | Meaning               |
|------------|-----------------------|
| subtitle   | Any character string. |

**Remarks:**

1. SUBTITLE appearing under a SUBCASE command will appear in the output for that subcase only.
2. SUBTITLE appearing before all SUBCASE commands will appear in the output for all subcases except those in Remark 1.
3. If no SUBTITLE command is present, the subtitle line will be blank.
4. The subtitle also appears on plotter output.

**SUPER****Superelement Subcase Assignment**

Assigns a subcase(s) to a superelement or set of superelements.

Format:

$$SUPER = \left\{ \begin{array}{c} ALL \\ \left\{ \begin{array}{c} n \\ i \end{array} \right\} [ , l ] \end{array} \right\}$$

Examples:

SUPER=17, 3

SUPER=15

SUPER=ALL

| Descriptor | Meaning  |
|------------|--|
| i          | Superelement identification number (Integer > 0).  |
| ALL        | The subcase is assigned to all superelements and all loading conditions (Default).   |
| n          | Set identification number of a previously appearing SET command. The subcase is assigned to all superelements with identification numbers that appear on this SET command (Integer > 0). |
| l          | Load sequence number (Integer > 0; Default=1).   |

Remarks:

1. All subcases with requests for specific superelement(s) must contain the SUPER command. If no SUPER command is specified in the Case Control Section, then all subcases will be assigned to all superelements; i.e., SUPER=ALL is the default.
2. All subcases associated with superelements must precede those for the residual structure except when SUPER=ALL or SUPER=n and the selected set includes the residual structure.
3. The load sequence number is only used in static analysis and frequency response analysis when there are multiple loading conditions. Also, the residual structure must have a subcase specified for each unique load condition. This is required because the number of residual structure subcases is used to determine the number of load conditions for all superelements.
4. The load sequence number is associated with the order of the subcases for the residual structure; i.e., the third loading condition is associated with the third subcase for the residual structure.
5. Subcases are required for superelements when there is a load, constraint, or output request.
6. If a set is referenced by n, then the SET identification number must be unique with respect to any superelement identification numbers. In addition, the same sets must be used for all loading conditions.

7. If the ALL option is used, it must be used for all loading conditions.
8. If there are no superelements in the model then the SUPER command will be ignored except in 3-step external superelement data recovery restarts in SOL 400. See Remarks 9 and 10 under the EXTDRIN Case Control command description.

## SUPPORT1

### Fictitious Support Set Selection

Selects the fictitious support set (SUPPORT1 or SUPPORT6 entries only) to be applied to the model.

**Format:**

SUPPORT1=n

**Examples:**

SUPPORT1=15

SUP0=4

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of fictitious support set defined on the SUPPORT1 or SUPPORT6 Bulk Data entries (Integer > 0). |

**Remarks:**

1. SUPPORT1 or SUPPORT6 Bulk Data entries will not be used unless selected in the Case Control Section by the SUPPORT1 command.
2. SUPPORT entries will be applied in all subcases.
3. For SOL 600, Case Control command SUPPORT1 must reference a SUPPORT6 Bulk Data entry with ID = N.

**SVECTOR****Solution Set Eigenvector Output Request**

Requests the form and type of solution set eigenvector output.

**Format:**

$$SVECTOR(\begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}) = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Examples:**

SVECTOR=ALL

SVECTOR (PUNCH) =NONE

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |  |
|------|--|
| ALL  | Displacements for all points (modes) will be output.   |
| NONE | Displacements for no points (modes) will be output.  |
| n    | Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. SVECTOR=NONE overrides an overall output request.
2. Output will be presented as a tabular listing of grid points for each eigenvector.

**SVELOCITY****Solution Set Velocity Output Request**

Requests the form and type of solution set velocity output.

**Format:**

$$\text{SVELOCITY}\left[\left(\begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix}\right), \text{PRINT}, \text{PUNCH}, \left[\begin{matrix} \text{REAL or IMAG} \\ \text{PHASE} \end{matrix}\right]\right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

SVELOCITY=5

SVELOCITY(SORT2, PUNCH, PRINT, PHASE)=ALL

| Descriptor   | Meaning   |
|--------------|---|
| SORT1        | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.                               |
| SORT2        | Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).  |
| PRINT        | The printer will be the output medium.  |
| PUNCH        | The punch file will be the output medium.   |
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.  |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.  |
| ALL          | Velocity for all solution points (modes) will be output.  |
| NONE         | Velocity for no solution points (modes) will be output.   |
| n            | Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output (Integer > 0). |

**Remarks:**

1. Velocity output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis, and is discussed in Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SVELOCITY=NONE overrides an overall output request.

**SYM** Symmetry Subcase Delimiter

Delimits and identifies a symmetry subcase.

**Format:**

SYM=n

**Example:**

SYM=123

| Descriptor | Meaning                                      |
|------------|--|
| n          | Subcase identification number (Integer > 0). |

**Remarks:**

1. The subcase identification number n must be greater than all previous subcase identification numbers.
2. Plot commands should refer to n.
3. Overall output commands will not propagate into a SYM subcase (i.e., any output desired must be requested within the subcase).
4. SYM may only be used in statics or inertia relief problems.

**SYMCOM**

## Symmetry Combination Subcase Delimiter

Delimits and identifies a symmetry combination subcase.

**Format:**

SYMCOM=n

**Example:**

SYMCOM=123

| Descriptor | Meaning                                      |
|------------|--|
| n          | Subcase identification number (Integer > 2). |

**Remarks:**

1. The subcase identification number n must be greater than all previous subcase identification numbers.
2. SYMCOM may only be used in statics problems.
3. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SYMCOM by use of a TEMP(LOAD) command, or the element deformations by a DEFORM command.
4. An alternate command is the SUBCOM command.
5. SYMCOMs may be specified in superelement analysis with the following recommendations:
  - a. For each superelement, specify its SUBCASEs consecutively, directly followed by its SYMCOM(s).
  - b. Specify a SUPER command with a new load sequence number under each SYMCOM command.

The following example represents a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SYMCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SYMSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SYMCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SYMSEQ=1.,1.
```

**SYMSEQ****Symmetry Sequence Coefficients**

Specifies the coefficients for combining symmetry subcases into the total structure.

**Format:**

SYMSEQ=R1 [,R2,R3,..., Rn]

**Example:**

SYMSEQ=1.0, -2.0, 3.0, 4.0

| Descriptor | Meaning   |
|------------|---|
| Ri         | Coefficients of the previously occurring n SYM subcases. (Real) |

**Remarks:**

1. SYMSEQ may only appear after a SYMCOM command.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. SYMSEQ may only be used in static analysis or inertia relief.
4. Ri is limited to a maximum of 200 numbers.

**TACCELERATION**

Acceleration Output Request for trim components, TRMC

Requests the form and type of acceleration output for TRMC.

**Format:**

$$TACCELERATION \left[ \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Example:**

```
SET 20 = 3/103, 5/0, 12/ALL $
TACCELERATION=ALL
TACC (REAL, PUNCH, PRINT)=20
TACCE=20
TACCELE (SORT2, PRINT)=20
```

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of grid points for each load and frequency. |                   |                      |
| SORT2            | Output will be presented as a tabular listing of frequency for each grid point.           |                   |                      |
|                  | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |
| ALL          | TRMC acceleration for all points of all TRMCs will be output. See Remarks 2.   |

| Descriptor | Meaning  |
|------------|--|
| NONE       | No TRMC acceleration will be output.   |
| n          | Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12. |

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TDISP=NONE suppresses the generation of TRMC acceleration output.
4. TACCELERATION is supported only with PARAM,TRMBIM,PHYSICAL
5. TACCELERATION is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TACCELERATION is available for output only and cannot be utilized as design response.
6. For restart PEM job, TACCE request must remain the same as cold start PEM job.

**TDISPLACEMENT**

Displacement Output Request for trim components, TRMC

Requests the form and type of displacement output for TRMC.

**Format:**

$$T DISPLACEMENT \left[ \begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT & PUNCH \\ PLOT \end{bmatrix} \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

**Example:**

```
SET 20 = 3/103, 5/0, 12/ALL $
T DISPLACEMENT=ALL
T DISP(REAL, PUNCH, PRINT)=20
T DISP=20
T DISPLACE(SORT2, PRINT)=20
```

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of grid points for each load and frequency. |                   |                      |
| SORT2            | Output will be presented as a tabular listing of frequency for each grid point.           |                   |                      |
|                  | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |
| ALL          | TRMC displacement for all points of all TRMCs will be output. See Remarks 2.   |

| Descriptor | Meaning  |
|------------|--|
| NONE       | No TRMC displacement will be output.   |
| n          | Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12. |

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TDISP=NONE suppresses the generation of TRMC acceleration output.
4. TDISPLACEMENT is supported only with PARAM,TRMBIM,PHYSICAL
5. TDISPLACEMENT is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TDISPLACEMENT is available for output only and cannot be utilized as design response.
6. For each grid in a TRMC, it has 3 or 6 DOFs for solid phase and 1 DOF for fluid phase. TDISPLAMENT output results for both solid phase DOFs and fluid phase DOF.
7. The 3 DOFs of solid phase takes T1, T2 and T3 locations of a regular 6 DOFs structural grid and set R1, R2 and R3 locations to 0.0. The 1 DOF of fluid phase is reported as acoustic pressure.
8. For restart PEM job, TDISP request must remain the same as cold start PEM job.

**TEMPERATURE****Temperature Set Selection**

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Format:

$$\text{TEMPERATURE} \left[ \begin{array}{c} \text{INITIAL} \\ \text{MATERIAL} \\ \text{LOAD} \\ \text{BOTH} \end{array} \right] \left[ , \text{HSUBCASE} = i \left[ , \text{HSTEP} = j \left[ , \text{HTIME} = \left[ \begin{array}{c} t \\ \text{ALL} \end{array} \right] \right] \right] \right], \text{VERIFY} = \left[ \begin{array}{c} \text{GRID} \\ \text{ELEMENT} \\ \text{BOTH} \\ \text{NONE} \end{array} \right]$$

Examples:

`TEMPERATURE (LOAD) =15`

`TEMPERATURE (MATERIAL) =7`

`TEMPERATURE=7`

`TEMPERATURE (LOAD, HSUBCASE=20, HTIME=12.0) =39`

| Descriptor | Meaning  |
|------------|--|
| MATERIAL   | The selected temperature set will be used to determine temperature-dependent material properties indicated on MATTi Bulk Data entries. See Remarks 6., 7., and 8.              |
| LOAD       | The selected temperature set will be used to determine an equivalent static load and to update material properties in a nonlinear analysis. See Remarks 2., 5., 6., 7. and 14. |
| BOTH       | Both MATERIAL and LOAD will use the same temperature set.  |
| n          | Set identification number of TEMP, TEMPD, TEMPP1,TEMPB3, TEMPRB, TEMPF, or TEMPAX Bulk Data entries (Integer > 0).   |
| INITIAL    | The selected temperature table will be used to determine initial temperature distribution in nonlinear static analysis. See Remarks 4., 6., 7., 8., 9., and 12.                |
| HSUBCASE   | Specifies a SUBCASE executed in the selected thermal job. See Remark 14.   |
| i          | Identification number of a SUBCASE executed in the selected thermal job. (Integer $\geq 0$ , Default = 0 is the first SUBCASE) See Remark 14.                                  |
| HSTEP      | Specifies a STEP executed in the selected thermal job. See Remark 14.  |
| j          | Identification number of a STEP executed in the selected thermal job. (Integer $\geq 0$ , Default = 0 is the first STEP) See Remark 14.  |
| HTIME      | Specifies the time of a time step executed in the selected nonlinear transient thermal job. See Remark 14.   |
| t          | Time of a time step executed in the selected nonlinear transient thermal job. (Real $\geq 0.0$ , Default is the last time of the specified SUBCASE and/or STEP) See Remark 14. |

| Descriptor | Meaning   |
|------------|---|
| ALL        | Selects all time steps executed in the selected nonlinear transient thermal job. See Remark 14. |
| VERIFY     | Output temperature verification data output requested   |
| GRID       | Output grid temperature verification data   |
| ELEMENT    | Output element temperature verification data  |
| BOTH       | Output both grid and element temperature verification data                                      |
| NONE       | Do not output any verification data   |

### Remarks:

1. In linear analysis, only one TEMP(MATE) may be made in any problem and should be specified above the subcase level. If there are multiple subcase and TEMP(MATE) is not above the first subcase, then it must appear in the last subcase or it will be ignored. See also Remarks 6. and 7.
2. The total load applied will be the sum of external (LOAD command), thermal (TEMP(LOAD) command), element deformation (DEFORM command), and constrained displacement (SPC command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.
4. INITIAL is used in steady state heat transfer analysis for conduction material properties, and provides starting values for iteration. In structural analysis, TEMP(INIT) is used to specify an initial temperature. It may appear above or in the first subcase. In SOL 400, it may appear above or in the first STEP.
5. In superelement data recovery restarts, TEMPERATURE(LOAD) requests must be respecified in the Case Control Section.
6. In linear static analysis, temperature strains are calculated by

$$\epsilon_T = A(T_o) \cdot (T - T_o)$$

where  $A(T_o)$  is the thermal expansion coefficient defined on the MATi Bulk Data entries,  $T$  is the load temperature defined with TEMPERATURE(LOAD), and  $T_o$  is the initial temperature defined as follows. The following rules apply for TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), and TREF on the MATi entries:

- a. If TEMPERATURE(INITIAL) and TREF are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties.
- b. If TEMPERATURE(MATERIAL) and TREF are specified, then TREF will be used as the initial temperature in calculating the load and the TEMPERATURE(MATERIAL) set will be used for the calculation of material properties.
- c. If neither TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), nor TEMPERATURE(BOTH) is present, TREF will be used to calculate both the load and the material properties and will be obtained from the MATi entry. The MATTi is not used in this case.

7. In nonlinear static analysis, temperature strains are calculated with

$$\epsilon_T = A(T) \cdot (T - TREF) - A(T_o) \cdot (T_o - TREF)$$

where  $A(T)$  is the thermal expansion coefficient defined on the MATi Bulk Data entries,  $T$  is the load temperature defined with TEMPERATURE(LOAD), and  $T_o$  is the initial temperature defined with TEMPERATURE(INITIAL). The following rules apply:

- a. The specification of TEMPERATURE(MATERIAL) or TEMPERATURE(BOTH) will cause a fatal error.
- b. If a subcase does not contain a TEMPERATURE(LOAD) request, then the thermal load set will default to the TEMPERATURE(INITIAL) set.
- c. TEMPERATURE(LOAD) will also cause the update of temperature-dependent material properties due to the temperatures selected in the thermal load set. Temperature-dependent material properties are specified with MATi, MATTi, MATS1, and/or TABLEST Bulk Data entries.
- d. If TREF and TEMPERATURE(INITIAL) are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties. Both are used in the definition of thermal strain.

For SOL 600, TREF and TEMP(INIT) must be consistent (the same values) or unexpected results may occur.

- 8. TEMPERATURE(MATERIAL) and TEMPERATURE(INITIAL) cannot be specified simultaneously in the same run.
- 9. TEMP(INIT) is not used with TEMPAX.
- 10. Temperature loads cause incorrect element forces and stresses in all the dynamic analysis except SOL 400 nonlinear transient analysis with the elements having nonlinear capability.

It should be noted that:

- a. For nonlinear dynamics analysis with SOL 400, TEMPERATURE/THERMAL load can't be applied by TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data), this load should be applied by TEMP(Case Control) and TTEMP(Bulk Data);
- b. For pure linear analysis with SOL 400, the TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data) has to be used to apply the temperature load.
- 11. In linear analysis, TEMPERATURE(MATERIAL) is not supported for hyperelastic elements (MATHP). TEMP(INIT) must be placed above the subcase level, and TEMP(LOAD) placed within the subcase.
- 12. For layered composites, neither the TREF specified on the material entries, nor TEMP(INIT) nor TEMP(MATE) are used to determine ply reference temperature. The TREF on the PCOMP or PCOMPG entries is used for all plies of the element. This is true for both linear and nonlinear analysis. If TEMP(INIT) is defined, the TREF on the PCOMP or PCOMPG entries is only used for determining the material properties and TEMP(INIT) is used for thermal strains. If TEMP(INIT) is not defined, the TREF on the PCOMP or PCOMPG entries is used for determining the material properties and thermal strains.

13. For SOL 600, in a thermal stress analysis where the temperatures were produced in a previous SOL 600 simulation, the use of MCHSTAT is preferred.
14. For TEMPERATURE(LOAD) requests in SOL 400, HSUBCASE, HSTEP, and HTIME are used to retrieve the temperature results from an existing thermal database. This feature allows user to select either steady state or transient thermal results for nonlinear structural analysis, with the flexibility of different time steps and dissimilar mesh sizes between thermal and structural runs. The following rules apply for using this capability:
  - HSUBCASE, HSTEP, and HTIME keywords must follow a LOAD keyword.
  - Although all three keywords have default values, at least one keyword must exist to apply this uncoupled multi-physics feature in analysis.
  - HTIME=ALL is used in nonlinear transient structural analysis to perform real time temperature interpolations. In this case, the nodal temperatures of nonlinear elements are updated at each time step. These temperatures are equal to the temperature results of the selected thermal database at current time.
  - The set IDs of TEMP(LOAD) must be different from the set IDs of TEMP(INIT).
  - To save temperature results of thermal analysis in MASTER nastran database, use the following command.

nastran thermal\_job\_name scratch=no

In addition, the user must specify the following Bulk Data entry:

PARAM,NLPACK,-1

for transient thermal models with number of time steps greater than the default NLPACK output time steps.

- The following File Management Statements are required in the current structural model to select the thermal database.

ASSIGN hrun='thermal\_job\_name.MASTER'

DBLOC DATABLK=(HEATDB) LOGI=hrun

- Contact body integrity is not maintained while doing the thermal mapping. While this would not be a restriction for bodies that are not physically in contact or in glued thermal contact, it could lead to undesirable temperature mapping for bodies that are in regular thermal contact with temperature gradients across the contact interface.

## TERMIN

## Conditions to Terminate in SOL 600

Selects a TERMIN Bulk Data entry which specifies criteria such that a SOL 600 analysis can be terminated, for example, if the displacement at a certain grid exceeds a specified value.

Format:

TERMIN=N

Example:

TERMIN=5

| Descriptor | Meaning  |
|------------|--|
| N          | ID of a matching TERMIN Bulk Data entry specifying the termination conditions for a particular analysis. |

Remarks:

1. This entry may only be used within subcases (it may not be placed above the first subcase entry). If there are no subcase entries, it may be placed anywhere in the Case Control Section.
2. Most SOL 600 analyses do not require TERMIN entries.
3. If some subcases have TERMIN entries and others do not, only those that do will check for termination conditions.
4. TERMIN criteria may be different for different subcases.

**TFL****Transfer Function Set Selection**

Selects the transfer function set(s) to be added to the direct input matrices.

**Format:**

TFL=n

**Example:**

TFL=77

TFL = 1, 25, 77

| Descriptor | Meaning   |
|------------|---|
| n          | Set identification of a TF Bulk Data entry (Integer > 0). |

**Remarks:**

1. Transfer functions will not be used unless selected in the Case Control Section.
2. Transfer functions are supported in dynamics problems only.
3. Transfer functions are described in the [MSC Nastran Dynamic Analysis User's Guide](#).
4. It is recommended that PARAM,AUTOSPC,NO be specified when using transfer functions. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
5. The transfer functions are additive if multiple TF values are referenced on the TFL command.

**THERMAL****Temperature Output Request**

Requests the form and type of temperature output.

**Format:**

$$\text{THERMAL} \left[ \left( \begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

**THERMAL=5**

**THER (PRINT, PUNCH)=ALL**

| Descriptor | Meaning  |
|------------|--|
| SORT1      | Output is presented as a tabular listing of point temperatures for each load or time step. |
| SORT2      | Output is presented as a tabular listing of loads or time steps for each.                  |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

**ALL** Temperatures for all points will be output.

**NONE** Temperatures for no points will be output.

**n** Set identification of a previously appearing SET command. Only temperatures of points with identification numbers that appear on this SET command will be output (Integer > 0).

**Remarks:**

1. The THERMAL output request is designed for use with the heat transfer option. The printed output will have temperature headings. The PUNCH option produces TEMP Bulk Data entries, and the SID on the entries will be the subcase number (=1 if no SUBCASES are specified).

2. SORT1 is the default in steady state heat transfer analysis. SORT2 is the default in transient heat transfer analysis.
3. In a transient heat transfer analysis, the SID on the punched TEMP Bulk Data entries equal the time step number.

**TITLE**      Output Title

Defines a character string to appear on the first heading line of each page of MSC Nastran printer output.

**Format:**

TITLE=title

**Example:**

TITLE=RIGHT WING, LOAD CASE 3.

| Descriptor | Meaning               |
|------------|-----------------------|
| title      | Any character string. |

**Remarks:**

1. If this command appears under a SUBCASE command, then the title appears in the output for that subcase only.
2. If this command appears before all SUBCASE commands, then the title is used in all subcases without a TITLE command.
3. If no TITLE command is present, then the title line will contain data and page numbers only.
4. The title also appears on plotter output.

**TRIM****Aerodynamic Trim Variable Constraint Selection**

Selects trim variable constraints in static aeroelastic response.

**Format:**

TRIM = n

**Example:**

TRIM=1

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of a TRIM Bulk Data entry (Integer > 0). |

**Remark:**

1. Aerodynamic extra points (trim variables) not constrained by a TRIM Bulk Data entry will be free during the static aeroelastic response solution.

**TRIMF**

## Trim Load Output Specification

Specifies options for the output of trim loads from a static aeroelastic analysis as FORCE/MOMENT Bulk Data entries.

**Format:**

**TRIMF**[([**UNIT** = *i*], [**LOADSET** = *n*], [**LARGE**], [**INERTIA**], [**APPLIED**], [**AIR**]

$$[**NOSUM**][**RIGID**],[**NOELASTIC**], [**QNORM**] )= \left\{ \begin{array}{l} **ALL** \\ n \end{array} \right\}$$

**Example:**

```
TRIMF (LOADSET=10001, LARGE) =ALL
TRIMF (UNIT=59, INERTIA, NOSUM) =1
```

| Descriptor       | Meaning   |
|------------------|---|
| <b>UNIT</b>      | Fortran unit to which data are written. (Optional; Default = 7) (punch file).   |
| <b>LOADSET</b>   | Load set id for output bulk data entries. If the TRIMF specification results in multiple load sets, then the defined ID will be used for the first and each subsequent load set has an ID incremented by 1. (Optional; Default = 1) |
| <b>LARGE</b>     | Write the output data in large field format (16 characters per field). The default is 8 characters per field.   |
| <b>INERTIA</b>   | Write out inertial loads as a separate load set. By default, the separate load set will not be written.   |
| <b>APPLIED</b>   | Write out applied loads as a separate load set. By default, the separate load set will not be written.  |
| <b>AIR</b>       | Write out aerodynamic loads as a separate load set. By default, the separate load set will not be written.  |
| <b>NOSUM</b>     | By default, the sum of the inertial, applied, and aerodynamic loads will be written as a separate load set. This option suppresses the writing of that set of loads.  |
| <b>RIGID</b>     | Write out rigid instances of the selected loads (Inertial, Applied, Air and/or Sum) as separate load sets. By default, the separate load set will not be written.   |
| <b>NOELASTIC</b> | By default, the sum of the rigid and elastic increment loads will be written as a separate load set. This option suppresses the writing of that set of loads.   |
| <b>QNORM</b>     | Normalize the load by the dynamic pressure used in the trim analysis. By default, the loads are not normalized.   |
| <b>ALL</b>       | Loads for all points will be output   |
| <b>n</b>         | Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output (Integer > 0)   |

**Remark:**

1. By default, the loads are written to the punch file (Fortran unit 7). If the user specifies an alternate Fortran unit number on the TRIMF entry, by default the loads will be written to a file name that is machine specific (i.e. 'fort.53' on many LINUX platforms). The user may connect the Fortran unit to a user-defined file name by using an ASSIGN entry in the FMS Section of the input file. For example:

```
ASSIGN USERFILE='load13.inc', STATUS=UNKNOWN, FORMATTED, UNIT=53
```

2. Up to eight loads sets are available: Rigid Inertial, Rigid Applied, Rigid Air, Rigid Sum and four more with the sum of the rigid and elastic increment. This table indicates how the describers invoke each of these sets:

|         | INERTIAL | APPLIED | AIR | SUM     |
|---------|----------|---------|-----|---------|
| RIGID   | O        | O       | O   | O       |
| ELASTIC | O        | O       | O   | DEFAULT |

where O = Optional. For a load to appear, both the row and column in the table above have to be set.

3. Care must be taken if LOADSET is specified in a run with multiple subcases. There are no checks that the load set IDs which are generated by one subcase are not also used for another subcase. For example, consider the following Case Control commands:

```
SUBCASE 1
  TRIM = 1
  TRIMF(RIGID) = ALL $
SUBCASE 2
  TRIM = 2
  TRIMF(LOADSET=2) = ALL
```

Subcase 1 will generate two load sets with set IDs 1 and 2. Subcase 2 will also output a load set ID 2.

4. The LOADSET option should not be specified above the subcase level when there are multiple subcases. If it is, each subcase will start numbering its load ID's from LOADSET.

**TRIMGRP****Selection of Trim Component(s)**

Selects a set of trim components for analysis.

**Format:**

$$TRIMGRP = \begin{bmatrix} sid \\ ALL \\ NONE \end{bmatrix}$$

**Example:**

TRIMGRP=101

| Descriptor | Meaning  |
|------------|--|
| sid        | Set identification of a previously appearing SET command or TRIMID. (Integer>0). |
| ALL        | All trim components present in the model are included in the analysis.           |
| NONE       | This is the default of TRIMGRP. No trim component is involved in the analysis.   |

**Remark:**

1. TRIMGRP is used to select a group of trim components for analysis, including the calculation of trim component boundary coupling matrices.:
2. PEM capability is available in SOL 108, SOL 111 and SOL 200 with analysis=mfreq.
3. SUBCASEs without TRIMGRP will have the equivalent effect of TRIMGRP=none even with the presence of 'BEGIN TRMC' in the deck.
4. To run PEM job with multiple processors, additional information is available in 'Running PEM jobs' section of [Using PEM Functions in MSC Nastran](#) in the *MSC Nastran Installation and Operations Guide*.
5. For restart PEM job, TRIMGRP must remain the same as cold start PEM job and no changes to any TRMC should be observed. Otherwise, the PEM job should be run as cold start.
6. Collapsed logic is introduced to improve performance of large PEM jobs satisfy following requirements:
  - a. Same TRIMGRP for all SUBCASEs and
  - b. same master frequencies for all TRMCs. The activation of collapsed logic is automatic if requirements are met. With collapsed logic activated, restart job is restricted to the same TRIMGRP as cold start.

**TSTEP****Transient Time Step Set Selection**

Selects integration and output time steps for linear or nonlinear transient analysis.

**Format:**

TSTEP=n

**Example:**

TSTEP=731

| Describer | Meaning  |
|-----------|--|
| n         | Set identification number of a TSTEP or Bulk Data entry (Integer > 0). |

**Remarks:**

1. A TSTEP entry must be selected to execute a linear transient analysis (SOLs 109 or 112) and for a nonlinear transient analysis (SOLs 129 and 159).
2. A entry must be selected in each subcase to execute a nonlinear transient problem.
3. For the application of time-dependent loads in modal frequency response analysis (SOLs 111 and 146), the TSTEP entry must be selected by the TSTEP command. The time-dependent loads will be recomputed in frequency domain by a Fourier transform.
4. In one subcase or STEP for SOL 400, users should only specify one of TSTEP, TSTEPNL or NLSTEP.

**TSTEPNL**

## Transient Time Step Set Selection for Nonlinear Analysis

See the description of the [TSTEP \(Case\), 629](#). (The TSTEP Case Control entry can be used to select a TSTEPNL Bulk Data entry, however, a Bulk Data TSTEP is completely different from a Bulk Data Entry TSTEPNL.)

**TSTRU**

## Temperature Set ID for a Structures Run

Defines a temperature set ID for a structures run based on a heat transfer subcase.

**Format:**

TSTRU=n

**Example:**

TSTRU=999

| Describer | Meaning  |
|-----------|--|
| n         | Set identification for use on TEMP(LOAD)=n or TEMP(INIT)=n |

**Remarks:**

1. TSTRU should be placed in a heat transfer subcase.
2. If TSTRU does not explicitly appear in the heat transfer subcase, it is defaulted to TSTRU=heat transfer subcase ID.
3. In a structures run, a temperature set generated from a heat transfer run will override an existing temperature set with identical set ID defined with TEMP, TEMPD, TEMPF, TEMPP1, TEMPRB, or any combination.
4. TSTRU may be placed in the first subcase of a PARAM,HEATSTAT,YES run.
5. TSTRUs may be placed in each subcase of an APPHEAT run. The associated structural analysis then requires the following:

```
ASSIGN heat_run='heat transfer job name.MASTER'
DBLOC DATABLK=(UG,EST,BGPDTS,CASECCR/CASEHEAT) LOGICAL=heat_run
```

6. Heat transfer runs and structural runs must have the same mesh. P-elements should also have the same geometry description. P-order between runs can be different.
7. For nonlinear heat transfer SOL 106 or SOL 153, the INOUT field on the NLPARM Bulk Data entry must be blank or NO if the results of the run are to be transferred to a linear structures run.

```
PARAM,NLHTLS,-1
```

The above parameter should be placed in the nonlinear heat run. This will place UG heat transfer on the database.

## TVELOCITY

Velocity Output Request for trim components, TRMC

Requests the form and type of velocity output for TRMC.

Format:

$$TVELOCITY \left[ \begin{pmatrix} SORT1 \\ SORT2 \end{pmatrix}, \begin{pmatrix} PRINT, PUNCH \\ PLOT \end{pmatrix} \begin{pmatrix} REAL \text{ or } IMAG \\ PHASE \end{pmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Example:

```
SET 20 = 3/103, 5/0, 12/ALL $
TVELOCITY=ALL
TVEL(REAL, PUNCH, PRINT)=20
TVELO=20
TVELOCI(SORT2, PRINT)=20
```

| Descriptor       | Meaning   |                   |                      |
|------------------|---|-------------------|----------------------|
| SORT1            | Output will be presented as a tabular listing of grid points for each load and frequency. |                   |                      |
| SORT2            | Output will be presented as a tabular listing of frequency for each grid point.           |                   |                      |
|                  | Printer File (.f06)   | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X   |                   | X*                   |
| PUNCH            |   | X                 | X*                   |
| PLOT             |   |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- |              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |
| ALL          | TRMC velocity for all points of all TRMCs will be output. See Remarks 2.   |

| Descriptor | Meaning  |
|------------|--|
| NONE       | No TRMC velocity will be output.   |
| n          | Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12. |

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TVELO=NONE suppresses the generation of TRMC velocity output.
4. TVELOCITY is supported only with PARAM,TRMBIM,PHYSICAL
5. TVELOCITY is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TVELOCITY is available for output only and can not be utilized as design response.
6. For restart PEM job, TVELO request must remain the same as cold start PEM job.

## UNGLUE

### Contact Body Unglue Selection

Selects the grids should use standard contact instead of glued contact in glued bodies in SOL 400.

**Format:**

UNGLUE=n

**Example:**

UNGLUE=10

| Descriptor | Meaning  |
|------------|--|
| n          | Set identification number of the UNGLUE Bulk Data entry (Integer > 0). |

**Remarks:**

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of UNGLUE Bulk Data entry is defined on BCONTACT Case Control command if applicable; however, the SID on UNGLUE Case Control command can overwrite it.

## VCCT

## Virtual Crack Closure Technique for SOLs 400/600 Analysis

Selects grid sets to be used for virtual crack closure analysis in SOL 600 and SOL 400.

**Format:**

VCCT=N

**Example:**

VCCT=0

VCCT=1

| Descriptor | Meaning   |
|------------|---|
| N          | ID of a matching Bulk Data VCCT entry specifying the crack. |

**Remarks:**

1. This entry can only be used in SOLs 400/600.
2. Different sets of cracks can be selected for different subcases using this option.
3. For SOL 600, N=0 may be entered above any subcases; then, the Bulk Data entry VCCT with ID=0 will be used in the Marc model definition section. The fracture mechanics calculations will be performed for all subcases. Otherwise, if N > 0, the matching Bulk Data entry VCCT will be used in Marc's history definition section for the applicable subcase, and all subsequent subcases, until a new VCCT is activated.

## VECTOR

## Displacement Output Request

Requests the form and type of displacement vector output.

See the description of the [DISPLACEMENT \(Case\), 298](#).

**VELOCITY****Velocity Output Request**

Requests the form and type of velocity vector output.

Format:

$$\begin{aligned}
 VELOCITY & \left[ \left( \begin{matrix} SORT1 \\ SORT2 \end{matrix} \right), \left[ \begin{matrix} PRINT, PUNCH \\ PLOT \end{matrix} \right], \left[ \begin{matrix} REAL \text{ or } IMAG \\ PHASE \end{matrix} \right], \left[ \begin{matrix} PSDF, ATOC, CRMS \\ \text{or RALL} \end{matrix} \right], \right. \\
 & \quad \left. \left[ \begin{matrix} RPPRINT \\ NORPRINT, RPUNCH \end{matrix} \right], [CID] \right] \\
 & = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

Examples:

VELOCITY=5

VELOCITY(SORT2, PHASE, PUNCH)=ALL

VELOCITY(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20

VELOCITY(PRINT, RALL, NORPRINT)=ALL

| Descriptor | Meaning   |
|------------|---|
| SORT1      | Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. |
| SORT2      | Output will be presented as a tabular listing of frequency or time for each grid point.   |

|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
|------------------|---------------------|-------------------|----------------------|
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|              |  |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE        | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.                             |

| Descriptor | Meaning  |
|------------|--|
| PSDF       | Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5. |
| ATOC       | Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.        |
| CRMS       | Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.                       |
| RALL       | Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.                                    |
| RPRINT     | Writes random analysis results in the print file (Default).  |
| NORPRINT   | Disables the writing of random analysis results in the print file.   |
| RPUNCH     | Writes random analysis results in the punch file.  |
| CID        | Request to print output coordinate system ID in printed output (.f06) file.  |
| ALL        | Velocity for all solution points will be output.   |
| NONE       | Velocity for no solution points will be output.  |
| n          | Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output (Integer > 0).  |

**Remarks:**

1. Velocity output is only available for transient and frequency response problems.
2. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.
3. VELOCITY=NONE overrides an overall output request.
4. Velocity results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
5. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
6. Note that the CID keyword affects only grid point related output, such as DISP, VELO, ACCEL, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid point-related output request anywhere in the Case Control Section to turn on the printing algorithm.

**VINTENSITY**

Vibration Intensity, VI, Output Request

Request output of vibration intensity for structural elements in SOLs 108 and 111 only.

Format:

$$\text{VINTENSITY} \left( \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

`VINTENSITY = ALL`

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|      |   |
|------|---|
| ALL  | Vibration Intensities will be computed for all supported structural elements.   |
| n    | Set identification of a previously defined set of structural elements. Vibration Intensities will be computed for the structural elements in this set only. |
| NONE | Vibration intensity will not be processed.  |

Remarks:

1. VINTENSITY = NONE overrides an overall request.
2. This Case Control command can be used in SOL 108 and SOL 111 only
3. Structural element types covered for VI computations are BAR, BEAM, QUAD4, TRIA3, QUADR, TRIAR, QUAD8, TRIA6, HEXA, PENTA and TETRA.
4. For BAR/BEAM element types, VI is computed in element axial direction using following equation

$$VI_x = \text{Real}[-(F_x v_x^* + V_1 v_y^* + V_2 v_z^* + T \omega_x^* - M_2 \omega_y^* + M_1 \omega_z^*)] / \text{Area},$$

Where  $F_x$  = axial force in element x direction,

$V_1$  = shear force in element y direction,

$V_2$  = shear force in element z direction,  
 $T$  = torsion about element x direction,  
 $M_2$  = Bending moment in element y direction,  
 $M_1$  = Bending moment in element z direction,  
 $v_i$  = Translational velocity about element i direction,  
 $\omega_i$  = Rotational velocity about element i direction,

'\*' superscript denotes complex conjugate of the complex value.  $VI_x$  is computed at both end of BAR/BEAM element and the average of ends VI is presented as element  $VI_x$ .

5. For 2D element types, such as QUAD4/TRIA3, VI is computed in element coordinate system using following equations

$$VI_x = \text{REAL}[-(V_x v_z - M_x \omega_y^* + M_{xy} \omega_x^* + F_x v_x^* + F_{xy} v_y^*) / \text{Area}_x, \\ VI_y = \text{REAL}[-(V_y v_z + M_y \omega_x^* - M_{xy} \omega_y^* + F_y v_y^* + F_{yx} v_x^*) / \text{Area}_y,$$

Where  $V_x, V_y$  = transverse shear forces,

$M_x, M_y$  = Bending moments,

$M_{xy}$  = twisting moment,

$F_x, F_y$  = Membrane forces,

$F_{xy}, F_{yx}$  = Membrane shear,

$\text{Area}_x, \text{Area}_y$  = Area in the corresponding element axes,

$v_i$  = Translational velocity about element i direction,

$\omega_i$  = Rotational velocity about element i direction,

'\*' superscript denotes complex conjugate of the complex value.  $VI_i, VI_i$  are computed in element coordinate system using average corner velocities and the element forces at the element center.

6. For 3D element types, VI is computed in basic coordinate system using following equations

$$VI_x = \text{REAL}[-(\sigma_x v_x + \tau_{xy} v_y + \tau_{xz} v_z)]$$

$$VI_y = \text{REAL}[-(\sigma_y v_y + \tau_{yx} v_x + \tau_{yz} v_z)]$$

$$VI_z = \text{REAL}[-(\sigma_z v_z + \tau_{zx} v_x + \tau_{zy} v_y)]$$

Where  $VI_x, VI_y, VI_z$  = VI in basic coordinate system,

$\sigma_x, \sigma_y, \sigma_z$  = Normal stresses,

$\tau_{xy}, \tau_{yz}, \tau_{xz}$  = Shear stresses,

$v_i$  = Translational velocity in basic i direction,

7. VI for all element types is presented in basic coordinate system in print and/or punch output.

**VUGRID****View Geometry Output for p-Version Analysis**

Requests output of view grid and view element entries used in p-version element data recovery.

**Format:**

$$VUGRID \left[ \begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right] = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

**Example:**

VUGRID (PRINT)=n

| Descriptor       | Meaning   |
|------------------|---|
| ALL              | All view element and grid entries will be output.   |
| n                | Set identification of a previously appearing SET command. Only those p-version elements with identification numbers that appear on this SET command will be output (Integer > 0). |
| PRINT or (blank) | Printer File (.f06)   |
| PUNCH            | Punch File (.pch)   |
| PLOT             | Plot File (.op2/.h5)  |

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

**Remarks:**

1. VUGRID is processed only when an analysis with p-version elements is requested.
2. Only one VUGRID command per analysis is allowed.
3. The VUGRID command is used only for output control, and does not in anyway affect the p-version analysis.
4. See parameters VUHEXA, VUTETRA, and VUPENTA in [Parameters, 783](#), for renaming element entries.
5. See parameters VUELJUMP and VUGJUMP in [Parameters, 783](#), for numbering of view grid and view element entries.

**WEIGHTCHECK****Rigid Body Mass Reduction Check**

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass in the g-set.

Format:

$$\text{WEIGHTCHECK} \left[ \begin{array}{l} \left( \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right) \text{PUNCH}, \text{SET} = \left\{ \begin{array}{l} \text{G}, \text{N}, \text{N} + \text{AUTOSPC}, \text{F}, \text{A}, \text{V} \\ \text{ALL} \end{array} \right\} \\ \text{GRID} = \text{gid}, \text{CGI} = \left[ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], \left[ \begin{array}{l} \text{WEIGHT} \\ \text{MASS} \end{array} \right] \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

Examples:

WEIGHTCHECK=YES

WEIGHTCHECK(GRID=12, SET=(G, N, A), MASS)=YES

| Descriptor  | Meaning  |
|-------------|--|
| PRINT       | Write output to the print file (Default).  |
| NOPRINT     | Do not write output to the print file.   |
| PUNCH       | Write output to the punch file.  |
| SET         | Selects degree of freedom set(s) (Default SET=G).  |
| gid         | Reference grid point for the calculation of rigid body motion. The default is the origin of the basic coordinate system.         |
| CGI         | For $\text{SET} \neq \text{G}$ , CGI = YES requests output of center of gravity and mass moments of inertia (Default: CGI = NO). |
| WEIGHT/MASS | Selects output in units of weight or mass (Default = WEIGHT).  |

Remarks:

1. WEIGHTCHECK must be specified above the subcase level.
2. For SET=N, N+AUTOSPC, F, or A, the WEIGHTCHECK command also outputs a percentage loss or gain in the reduced rigid body mass matrix (e.g., MAA) as compared to the g-set rigid body mass matrix (e.g., MGG). G must also be requested to obtain this comparison; e.g., WEIGHTCHECK(SET=(G,A))=YES.
3. SET=N+AUTOSPC uses the mass matrix for the n-set with the rows corresponding to degrees of freedom constrained by the PARAM, AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.
4. WEIGHTCHECK is available in all SOLs. However for the residual structure in SOLs 101, 105, 114, and 116, because no mass reduction is performed, only WEIGHTCHECK(SET=J) is available. The 'J' set does not include upstream superelements.

5. If Lagrange multipliers are present via RIGID=LAGRAN or LGELIM then for degree-of-freedom sets N, N+AUTOSPC, F, and A the check will be performed on degree-of-freedom sets NL, NL+AUTOSPC, FL, and AL. The output will also be labeled accordingly.

**WETSENS**

## Sensitivity Wetted Grids for SOL 108/111

Select SOLUTION frequencies and RESPONSE DOFs for the generation of sensitivity for wetted grids.

Format:

$$WETSENS \left( \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix}, [THRESH = P], RESPONSE = r, \left[ SOLUTION = \begin{cases} ALL \\ self \end{cases} \right] \right),$$

$$[WETTED, SQWETT] = \begin{cases} ALL \\ n \\ NONE \end{cases}$$

Example:

```
SET 81 = 100.0, 120.0
SET 91 = 11240/T3, 4001/T1
SET 95 = 9000000 THRU 9000050
$
WETSENS (RESPONSE=91,solution=81,WETTED) = 95
```

| Descriptor       | Meaning             |                   |                      |
|------------------|---------------------|-------------------|----------------------|
|                  | Printer File (.f06) | Punch File (.pch) | Plot File (.op2/.h5) |
| PRINT or (blank) | X                   |                   | X*                   |
| PUNCH            |                     | X                 | X*                   |
| PLOT             |                     |                   | X*                   |

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

|          |  |
|----------|--|
| THRESH   | The magnitude of element sensitivity less than p will be suppressed in all output files: print, punch, plot, .op2, and .xdb. (Default = 0.0).          |
| RESPONSE | Adjoint load response will be computed for unit load applied at grid point components in SET r.  |
| SOLUTION | Frequency responses at these forcing frequencies, defined in setf, will be used for element sensitivity computation. (Default=all forcing frequencies) |
| WETTED   | Sensitivity for wetted grids will be computed and output.  |

| Descriptor | Meaning   |
|------------|---|
| SQWETT     | Squared sensitivity for wetted grids will be computed and output.   |
| ALL        | Sensitivities for all elements will be calculated.  |
| n          | Set identification number. Sensitivity for all elements specified on the SET n command will be calculated. The SET n command must be specified in the same subcase as the ELSSENS command, or above all subcases ( Integer > 0 ). The IDs in set n must be GID (grid ID). |
| NONE       | Elemental sensitivity will not be output.   |

**Remarks:**

1. Set r for RESPONSE on WETSENS is default to set r on ELSENS. If no ELSENS in the deck, set r for WETSENS must be provided.
2. The equations for various options of WETSENS

$$\text{WETSENS(WETTED)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r]$$

$$\text{WETSENS(SQWETT)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r] + [U_{\text{setf}}]^{*t} [\text{AGG}]^* [U_r]^*$$

where  $[U_{\text{setf}}]$  is the displacement of SOLUTION

$[U_r]$  is the displacement of RESPONSE

$[\text{AGG}]$  is Fluid/Structure Coupling matrix

superscript \* means complex conjugate of the term.

## Case Control Applicability Tables

The following tables describe the applicability of Case Control commands to Solution Sequences:

|  |   |
|--|---|
| <a href="#">Table 5-3</a> and<br><a href="#">Table 5-4</a> | SOLs (101 through 200) -- Subcase Definition, Superelement Control, and Auxiliary Model Control |
| <a href="#">Table 5-5</a> and<br><a href="#">Table 5-6</a> | SOLs (101 through 200) -- Data Selection  |
| <a href="#">Table 5-7</a> and<br><a href="#">Table 5-8</a> | SOLs (101 through 200) -- Output Selection  |

Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

| Command Name                     | Solution Number |     |     |     |     |     |     |     |     |     |
|----------------------------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                                  | 101             | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| ADACT                            | X               | X   |     |     |     |     |     |     |     |     |
| ANALYSIS                         |                 |     |     |     |     |     |     |     |     |     |
| AUXCAS                           |                 |     |     |     |     |     |     |     |     |     |
| AUXMODEL                         |                 |     |     |     |     |     |     |     |     |     |
| BEGIN BULK                       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| CAMPBLL                          |                 |     |     |     | X   |     |     |     |     |     |
| EXTDRIN                          | X               | X   |     |     | X   | X   | X   | X   | X   | X   |
| EXTDROUT                         | X               | X   |     |     | X   | X   | X   | X   | X   | X   |
| EXTSEOUT                         | X               | X   |     |     | X   | X   | X   | X   | X   | X   |
| MASTER                           | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| MCFRACTION                       |                 |     |     |     |     |     |     | X   | X   | X   |
| MODES                            |                 | X   |     |     |     |     |     |     |     |     |
| OUTPUT(blank)                    | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| OUTPUT(PLOT)                     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| OUTPUT (POST) or SETS DEFINITION | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| OUTPUT (XYPLOT)                  | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| REPCASE                          | X               | X   |     |     |     |     |     |     |     |     |
| SEALL                            | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SEDR                             | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SEDV                             |                 |     |     |     |     |     |     |     |     |     |
| SEEXCLUD                         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SEFINAL                          | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SEKRR                            | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SELG                             | X               |     | X   | X   |     | X   | X   |     | X   | X   |
| SELR                             | X               |     | X   | X   |     | X   | X   |     | X   | X   |
| SEMR                             | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SEMR                             |                 | X   | X   |     | X   | X   | X   | X   | X   | X   |
| SERE                             |                 |     |     |     |     |     |     |     |     |     |

**Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control**

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101             | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| STOCHASTICS  | X               | X   |     |     |     |     |     |     |     |     |
| SUBCASE      | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SUBCOM       | X               |     |     |     |     |     |     |     |     |     |
| SUBSEQ       | X               |     |     |     |     |     |     |     |     |     |
| SUPER        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SYM          | X               |     |     |     |     |     |     |     |     |     |
| SYMCOM       | X               |     |     |     |     |     |     |     |     |     |
| SYMSEQ       | X               |     |     |     |     |     |     |     |     |     |

**Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control**

| Command Name                     | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|----------------------------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                                  | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| ADACT                            |                 |     |     |     |     |     |     |     |     |     |     |     |
| ANALYSIS                         |                 |     |     |     |     |     |     |     | X   |     | X   |     |
| AUXCAS                           |                 |     |     |     |     |     |     |     |     |     | X   |     |
| AUXMODEL                         |                 |     |     |     |     |     |     |     |     |     | X   |     |
| BEGIN BULK                       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| EXTDRIN                          |                 |     |     |     |     |     |     |     |     |     |     | X   |
| EXTDROUT                         |                 |     |     |     |     |     |     |     |     |     |     | X   |
| EXTSEOUT                         |                 |     |     |     |     |     |     |     |     |     |     | X   |
| MASTER                           | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| MODES                            |                 | X   | X   |     |     |     |     |     |     |     |     | X   |
| OUTPUT(blank)                    | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| OUTPUT(PLOT)                     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| OUTPUT (POST) or SETS DEFINITION | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| OUTPUT(XYPLOT)                   | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| REPCASE                          |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SEALL                            | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |

Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| SEDR         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SEDV         |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SEEXCLUD     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SEFINAL      | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SEKRR        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SELG         | X               |     | X   | X   | X   | X   |     | X   | X   | X   | X   |     |
| SELRL        | X               |     | X   | X   | X   | X   |     | X   | X   | X   | X   |     |
| SEMR         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SEMR         |                 | X   | X   | X   | X   | X   | X   | X   |     | X   | X   |     |
| SERE         |                 |     |     |     |     |     |     |     |     |     |     | X   |
| STOCHASTICS  |                 |     |     |     |     |     |     |     |     |     |     | X X |
| SUBCASE      | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SUBCOM       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SUBSEQ       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SUPER        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SYM          |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SYMCOM       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SYMSEQ       |                 |     |     |     |     |     |     |     |     |     |     | X   |

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection

| Command Name | Structured Solution Number |     |     |     |     |     |     |     |     |     |  |
|--------------|----------------------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|--|
|              | 101                        | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |  |
| ADAPT        | X                          | X   |     |     |     |     |     |     |     |     |  |
| AUTOSPC      | X                          | X   | X   |     | X   | X   | X   | X   | X   | X   |  |
| AXISYMME     | X                          | X   | X   |     | X   | X   | X   | X   | X   | X   |  |
| B2GG         | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |  |
| B2PP         |                            |     |     |     | X   | X   | X   | X   | X   | X   |  |
| BC           |                            | X   |     |     |     |     |     |     |     |     |  |
| CLOAD        |                            |     |     | X   |     |     |     |     |     |     |  |
| CMETHOD      |                            |     |     |     | X   |     |     |     | X   |     |  |

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

| Command Name | Structured Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|----------------------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101                        | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| DEFORM       | X                          |     | X   |     |     |     |     |     |     |     |
| DESGLB       |                            |     |     |     |     |     |     |     |     |     |
| DESOBJ       |                            |     |     |     |     |     |     |     |     |     |
| DESSUB       |                            |     |     |     |     |     |     |     |     |     |
| DLOAD        |                            |     |     |     | X   | X   | X   | X   | X   | X   |
| DSYM         |                            |     |     |     |     |     |     |     |     |     |
| FMETHOD      |                            |     |     |     |     |     |     |     |     |     |
| FREQUENC     |                            |     |     |     |     | X   |     |     | X   |     |
| GUST         |                            |     |     |     |     |     |     |     |     |     |
| HARMONIC     | X                          | X   | X   |     | X   | X   | X   | X   | X   | X   |
| IC           |                            |     |     |     |     |     | X   |     |     |     |
| K2GG         | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| K2PP         |                            |     |     |     | X   | X   | X   | X   | X   | X   |
| LOAD         | X                          |     | X   | X   |     | X   | X   |     | X   | X   |
| LOADSET      | X                          |     | X   | X   |     | X   | X   |     | X   | X   |
| M2GG         | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| M2PP         |                            |     |     |     | X   | X   | X   | X   | X   | X   |
| METHOD       |                            | X   |     | X   | X   | X   | X   | X   | X   | X   |
| MFLUID       |                            | X   |     |     | X   | X   | X   | X   | X   | X   |
| MODTRAK      |                            |     |     |     |     |     |     |     |     |     |
| MPC          | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| NLPARM       |                            |     |     |     | X   |     |     |     |     |     |
| NONLINEA     |                            |     |     |     |     |     | X   |     |     | X   |
| OMODES       |                            | X   | X   |     |     |     |     | X*  | X*  | X*  |
| P2G          | X                          |     | X   | X   |     | X   | X   |     | X   | X   |
| RANDOM       |                            |     |     |     |     | X   |     |     | X   |     |
| RESVEC       |                            | X   |     | X   | X   | X   | X   | X   | X   | X   |
| SDAMPING     |                            |     |     |     |     |     |     | X   | X   | X   |

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

| Command Name | Structured Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|----------------------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101                        | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| SDENSITY     | X                          | X   | X   |     |     |     |     |     |     |     |
| SMETHOD      | X                          |     |     | X   |     | X   |     |     | X   |     |
| SPC          | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| STATSUB*     | X                          | X   | X   |     | X   | X   | X   | X   | X   | X   |
| SUPPORT1     | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| TEMPER(INIT) |                            |     |     | X   |     |     |     |     |     |     |
| TEMPER(LOAD) | X                          |     | X   | X   |     | X   | X   |     | X   | X   |
| TEMPER(MATE) | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| TFL          |                            |     |     |     | X   | X   | X   | X   | X   | X   |
| TRIM         |                            |     |     |     |     |     |     |     |     |     |
| TRIMGRP      |                            |     |     |     |     |     |     |     | X   |     |
| TSTEP        |                            |     |     |     |     |     | X   |     |     | X   |
| WEIGHTCHECK  | X                          | X   | X   | X   | X   | X   | X   | X   | X   | X   |

\*If STATSUB is specified, then the Case Control commands that select static loads become applicable to the solution sequence supporting STATSUB.

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 |
| ADAPT        |                 |     |     |     |     |     |     |     |     |     |     |
| AUTOSPC      | X               | X   | X   | X   |     | X   | X   | X   |     | X   | X   |
| AXISYMME     |                 |     |     |     |     |     |     |     |     |     | X   |
| B2GG         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| B2PP         |                 |     |     | X   | X   |     | X   | X   |     | X   | X   |
| BC           |                 |     |     |     |     |     |     |     |     |     |     |
| CLOAD        |                 |     |     |     |     |     |     |     | X   |     |     |
| CMETHOD      |                 |     |     |     |     |     | X   |     |     |     |     |
| DEFORM       | X               |     | X   |     |     | X   |     |     |     |     | X   |
| DESGLB       |                 |     |     |     |     |     |     |     |     | X   |     |
| DESMOD       |                 |     |     |     |     |     |     |     |     |     | X   |

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| DESOBJ       |                 |     |     |     |     |     |     |     |     |     | X   |     |
| DESSUB       |                 |     |     |     |     |     |     |     |     |     | X   |     |
| DESVAR       |                 |     |     |     |     |     |     |     |     |     | X   |     |
| DLOAD        |                 |     |     | X   | X   |     | X   | X   |     | X   | X   |     |
| DSYM         | X               | X   | X   | X   |     |     |     |     |     |     |     |     |
| FMETHOD      |                 |     |     |     |     | X   |     |     |     |     | X   |     |
| FREQUENC     |                 |     |     | X   |     |     |     | X   |     |     | X   |     |
| GUST         |                 |     |     |     |     |     | X   | X   |     |     |     |     |
| HARMONIC     | X               | X   | X   | X   |     |     |     |     |     |     | X   |     |
| IC           |                 |     |     |     |     |     |     |     |     |     | X   |     |
| K2GG         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| K2PP         |                 |     |     | X   | X   |     | X   | X   |     | X   | X   |     |
| LOAD         | X               |     | X   | X   | X   | X   |     | X   | X   |     | X   |     |
| LOADSET      | X               |     | X   | X   | X   | X   |     | X   | X   | X   | X   |     |
| M2GG         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| M2PP         |                 |     |     | X   | X   |     | X   | X   |     | X   | X   |     |
| METHOD       |                 | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| MFLUID       |                 | X   | X   | X   | X   | X   | X   | X   |     |     | X   |     |
| MODTRAK      |                 |     |     |     |     |     |     |     |     |     |     |     |
| MPC          | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| NLBUCK       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| NLPARM       |                 |     |     |     |     |     |     |     | X   |     |     |     |
| NONLINEA     |                 |     |     |     | X   |     |     |     | X   |     |     |     |
| OMODES       |                 | X   |     |     |     |     |     |     |     |     |     | X   |
| P2G          | X               |     | X   | X   | X   | X   |     | X   | X   |     | X   |     |
| RANDOM       |                 |     |     | X   |     |     |     | X   |     |     |     |     |
| RESVEC       |                 | X   |     | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SDAMPING     |                 |     |     |     |     |     | X   | X   |     |     |     | X   |
| SMETHOD      |                 |     |     |     |     |     |     |     |     |     |     |     |
| SPC          | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| STATSUB      |                 |     |     | X   |     |     |     |     |     |     | X   |     |

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| SUPPORT1     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| TEMPER(INIT) |                 |     |     |     |     |     |     |     |     |     |     |     |
| TEMPER(LOAD) | X               |     |     | X   |     | X   |     | X   |     |     |     | X   |
| TEMPER(MATE) | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| TFL          |                 |     |     | X   | X   |     | X   | X   |     | X   | X   |     |
| TRIM         |                 |     |     |     |     | X   |     |     |     |     |     | X   |
| TRIMGRP      |                 |     |     |     |     |     |     |     |     |     |     | X   |
| TSTEP        |                 |     |     |     | X   |     |     | X   |     | X   | X   |     |
| WEIGHTCHECK  | X               | X   | X   | X   | X   | X   | X   | X   |     |     |     | X   |

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101             | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| ACCELERA     |                 | X   |     |     |     | X   | X   |     | X   | X   |
| ACFPMRESULT  |                 |     |     |     |     | X   |     |     |     | X   |
| ACPOWER      |                 |     |     |     |     | X   |     |     |     | X   |
| AEROF        |                 |     |     |     |     |     |     |     |     |     |
| APPRESSURE   |                 |     |     |     |     |     |     |     |     |     |
| BOUTPUT      |                 |     |     | X   |     |     |     |     |     |     |
| CMSENRGY     |                 | X   |     | X   |     |     |     | X   | X   | X   |
| DATAREC      | X               | X   |     |     |     | X   | X   |     | X   | X   |
| DISPLACE     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| DSAPRT       |                 |     |     |     |     |     |     |     |     |     |
| ECHO         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| EDE          |                 | X   |     |     | X   | X   | X   | X*  | X   | X   |
| EKE          |                 | X   |     |     | X   | X   | X   | X*  | X   | X   |
| ELSDCON      | X               |     | X   |     |     |     |     |     |     |     |
| ELSUM        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| ENTHALPY     |                 |     |     |     |     |     |     |     |     |     |
| ERP          |                 |     |     |     |     | X   |     |     | X   |     |

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

| Command Name      | Solution Number |     |     |     |     |     |                |     |     |                |
|-------------------|-----------------|-----|-----|-----|-----|-----|----------------|-----|-----|----------------|
|                   | 101             | 103 | 105 | 106 | 107 | 108 | 109            | 110 | 111 | 112            |
| ESE               | X               | X   | X   | X   | X   | X   | X              | X*  | X   | X              |
| FLUX              | X               |     |     |     |     |     |                |     |     |                |
| FORCE             | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| GPFORCE           | X               | X   | X   | X   |     |     | X <sup>+</sup> |     |     | X <sup>+</sup> |
| GPKE              |                 | X   |     |     |     |     |                | X*  | X*  | X*             |
| GPSDCON           | X               |     | X   |     |     |     |                |     |     |                |
| GPSTRAIN          | X               | X   |     | X   |     |     | X              |     |     | X              |
| GPSTRESS          | X               | X   |     | X   |     |     | X              |     |     | X              |
| GROUNDCHECK       | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| HARMONIC          | X               | X   | X   |     | X   | X   | X              | X   | X   | X              |
| HDOT              |                 |     |     |     |     |     |                |     |     |                |
| HOUTPUT           |                 |     |     |     |     |     |                |     |     |                |
| INTENSITY         |                 |     |     |     |     | X   |                |     |     | X              |
| LABEL             | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| LINE              | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| MAXLINES          | X               | X   | X   |     |     |     | X              |     |     | X              |
| MAXMIN (old form) | X               | X   | X   |     |     |     | X              |     |     | X              |
| MAXMIN (DEF)      | X               | X   | X   | X   |     |     | X              |     |     | X              |
| MPCFORCE          | X               | X   | X   |     | X   | X   | X              | X   | X   | X              |
| MPRES             |                 | X   |     |     | X   | X   | X              | X   | X   | X              |
| NLLOAD            |                 |     |     |     |     |     | X              |     |     | X              |
| NOUTPUT           |                 |     |     |     |     |     |                |     |     |                |
| OFREQUEN          |                 |     |     |     |     | X   |                |     |     | X              |
| OLOAD             | X               |     | X   | X   |     | X   | X              |     | X   | X              |
| OTIME             |                 |     |     |     |     |     | X              |     |     | X              |
| OUTRCV            | X               | X   |     |     |     | X   | X              |     |     | X              |
| PAGE              | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| PARTN             | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| PLOTID            | X               | X   | X   | X   | X   | X   | X              | X   | X   | X              |
| POST              | X               |     |     |     |     |     |                |     |     |                |
| PRESSURE          |                 | X   |     |     | X   | X   | X              |     |     |                |

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101             | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| SACCELER     |                 |     |     |     |     | X   | X   |     | X   | X   |

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 101             | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 |
| SDISPLAC     |                 | X   | X   |     | X   | X   | X   |     | X   | X   |
| SET          | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SKIP         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SPCFORCE     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| STRAIN       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| STRESS       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| STRFIELD     | X               | X   | X   |     |     |     | X   |     |     | X   |
| SUBTITLE     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| SURFACE      | X               | X   |     | X   |     |     | X   |     |     | X   |
| SVECTOR      |                 | X   | X   |     |     |     |     | X   | X   | X   |
| SVELOCITY    |                 |     |     |     |     | X   | X   |     | X   | X   |
| THERMAL      | X               |     |     |     |     |     |     |     |     |     |
| TITLE        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| VECTOR       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   |
| VELOCITY     |                 |     |     |     |     | X   | X   |     | X   | X   |
| VOLUME       | X               | X   |     | X   |     |     | X   |     |     | X   |
| VUGRID       | X               | X   |     |     |     | X   | X   |     | X   | X   |

<sup>\*</sup>For modal part of solution.<sup>†</sup>Forces limited to stiffness contributions.

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection

| Command Name    | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|-----------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                 | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| ACCELERA        |                 |     | X   | X   |     |     | X   |     |     | X   |     |     |
| ACFPMRESULT     |                 |     |     |     |     |     |     |     |     |     |     |     |
| ACPOWER         |                 |     |     |     |     |     |     |     |     |     |     |     |
| AEROF           |                 |     |     |     |     | X   | X   | X   |     |     | X   |     |
| APPRESSURE      |                 |     |     |     | X   |     |     |     |     |     | X   |     |
| BOUTPUT         |                 |     |     | X   |     |     |     |     | X   | X   |     |     |
| CMSENRGY        |                 |     |     |     |     | X   | X   |     |     |     | X   |     |
| DATAREC         |                 |     |     |     |     |     |     |     |     |     | X   |     |
| DISPLACE        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| DSAPRT          |                 |     |     |     |     |     |     |     |     |     |     | X   |
| ECHO            | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| EDE             |                 | X   |     | X   |     |     |     |     |     |     |     | X   |
| EKE             |                 | X   |     | X   |     |     |     |     |     |     |     | X   |
| ELSDCON         |                 |     |     |     |     | X   |     |     |     |     |     | X   |
| ELSUM           | X               | X   | X   | X   | X   | X   | X   | X   |     |     |     | X   |
| ENTHALPY        |                 |     |     |     |     |     |     |     |     |     | X   |     |
| ERP             |                 |     |     |     |     |     |     |     |     |     | X   | X   |
| ESE             | X               | X   | X   | X   |     | X   |     |     |     |     |     | X   |
| FLUX            | X               |     |     |     |     |     |     |     |     | X   | X   | X   |
| FORCE           | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| GPFORCE         | X               | X   | X   | X   |     |     |     |     |     |     |     | X   |
| GPKE            |                 | X   |     |     |     |     |     |     |     |     |     | X   |
| GPSDCON         | X               |     |     |     |     | X   |     |     | X   |     |     | X   |
| GPSTRAIN        | X               | X   | X   |     | X   |     |     | X   |     |     |     | X   |
| GPSTRESS        | X               | X   | X   |     | X   | X   |     |     |     |     | X   | X   |
| GROUNDCHEC<br>K | X               | X   | X   | X   | X   | X   | X   | X   |     |     |     | X   |
| HARMONY         | X               | X   | X   | X   |     |     |     |     |     |     |     | X   |
| HDOT            |                 |     |     |     |     |     |     |     |     | X   |     |     |
| HOUTPUT         | X               | X   | X   | X   |     |     |     |     |     |     |     |     |

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| INTENSITY    |                 |     |     |     |     |     |     |     |     |     |     |     |
| LABEL        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| LINE         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| MAXLINES     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| MAXMIN(DEF)  |                 |     |     |     |     |     |     |     |     |     |     | X   |
| MPCFORCE     | X               | X   | X   | X   |     | X   | X   | X   |     |     | X   |     |
| MPRES        |                 |     | X   |     |     | X   | X   | X   |     |     | X   |     |
| NLLOAD       |                 |     |     |     |     |     |     |     |     |     | X   |     |
| NLOPRM       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| NLSTEP       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| NOUTPUT      | X               | X   | X   | X   |     |     |     |     |     |     |     |     |
| OFREQUEN     |                 |     |     |     | X   |     |     |     | X   |     |     | X   |
| OLOAD        | X               |     | X   | X   | X   | X   |     |     | X   | X   | X   | X   |
| OTIME        | X               |     | X   | X   | X   | X   |     |     | X   | X   | X   | X   |
| OUTRCV       |                 |     |     |     |     |     |     |     |     |     |     |     |
| PAGE         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| PARTN        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| PLOTID       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| PRESSURE     |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SACCELER     |                 |     |     |     | X   | X   |     |     | X   |     |     | X   |
| SDISPLAC     |                 | X   |     |     | X   | X   |     | X   | X   |     | X   | X   |
| SET          | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SKIP         | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| SPCFORCE     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| STRAIN       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| STRESS       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| STRFIELD     | X               | X   | X   |     |     | X   |     |     |     |     |     | X   |
| SUBSET       |                 |     |     |     |     |     |     |     |     |     |     | X   |
| SUBTITLE     | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

| Command Name | Solution Number |     |     |     |     |     |     |     |     |     |     |     |
|--------------|-----------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|              | 114             | 115 | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 |
| SURFACE      | X               | X   | X   |     |     |     |     |     |     |     | X   |     |
| SVECTOR      |                 | X   | X   | X   |     |     | X   |     | X   |     | X   |     |
| SVELOCITY    |                 |     |     | X   | X   |     |     | X   |     |     | X   |     |
| TEMPERATURE  |                 |     |     |     |     |     |     |     |     |     |     |     |
| THERMAL      | X               |     |     |     |     |     |     |     |     | X   | X   | X   |
| TITLE        | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| TRIMF        |                 |     |     |     |     | X   |     |     |     |     |     | X   |
| VECTOR       | X               | X   | X   | X   | X   | X   | X   | X   | X   | X   | X   |     |
| VELOCITY     |                 |     |     | X   | X   |     |     | X   |     |     | X   |     |
| VOLUME       | X               | X   | X   |     |     |     |     |     |     |     | X   |     |
| VUGRID       |                 |     |     |     |     |     |     |     |     |     |     |     |

## X-Y PLOT Commands

The X-Y output request packet of the Case Control Section includes all commands between either OUTPUT(XYPLOT) or OUTPUT(XYOUT), and either BEGIN BULK or OUTPUT(PLOT). The remainder of this section describes the X-Y output commands.

A single set of plotted X-Y pairs is known as a curve. Curves are the entities to be plotted. The surface (paper, microfilm frame, etc.) on which one or more curves is plotted is known as a frame. Curves may be plotted on a whole frame, an upper-half frame, or a lower-half frame. Grid lines, tic marks, axes, axis labeling, and other graphic control options may be chosen by the user. The program will select defaults for parameters not selected by the user.

Only two commands are required for an X-Y output request.

1. Only one of OUTPUT(XYPLOT) or OUTPUT(XYOUT) at the beginning of the X-Y output command packet.
2. At least one of the commands XYPILOT, XYPEAK, XYPRINT, XYPUNCH, XYPAPILOT.

The commands OUTPUT(XYPLOT) and OUTPUT(XYOUT) are equivalent. If the X-Y output is to be printed and/or punched, a PLOTTER command is not required.

If only the required commands are used, the graphic control options will all assume default values. Curves using all default parameters have the following general characteristics.

1. Tic marks are drawn on all edges of the frame. Five spaces are provided on each edge of the frame.
2. All tic marks are labeled with their values.
3. Linear scales are used.
4. Scales are selected such that all points fall within the frame.
5. The plotter points are connected with straight lines.
6. The plotted points are not identified with symbols.

The above characteristics may be modified by inserting any of the parameter definition commands in the next section, ahead of the XY\_\_\_\_ command(s). The use of a parameter definition command sets the value of that parameter for all following command operation commands unless the CLEAR command is inserted. If grid lines are requested, they will be drawn at the locations of all tic marks that result from defaults or user request. The locations of tic marks (or grid lines) for logarithmic scales cannot be selected by the user. Values for logarithmic spacing are selected by the program. The values for the number of tic marks (or grid lines) per cycle depend on the number of logarithmic cycles required for the range of the plotted values.

The definition and rules for the X-Y output commands follow. The form of X-Y output commands differ in many instances from that of similar commands used in the OUTPUT(PLOT) section.

## X-Y Output Command Summary

|         | Commands Applied To All Curves   |
|---------|--|
| PLOTTER | Selects format of plot file for interpretation by plotter postprocessor. |
| CAMERA  | Selects plotter media.   |

| Commands Applied To All Curves              |   |
|---|---|
| PENSIZE                                     | Selects pen number.   |
| DENSITY                                     | Selects the line density for microfilm plotters only.                             |
| XPAPER                                      | Defines the size of the paper in x-direction.                                     |
| YPAPER                                      | Defines the size of the paper in y-direction.                                     |
| XMIN  | Specifies the minimum value on the x-axis.  |
| XMAX  | Specifies the maximum value on the x-axis.  |
| XLOG  | Selects logarithmic or linear x-axis.   |
| YAXIS                                       | Controls the plotting of the y-axis on all curves.                                |
| XINTERCEPT                                  | Specifies the location of the x-axis on the y-axis.                               |
| UPPER TICS                                  | Specifies how to draw tic marks on upper edge.                                    |
| LOWER TICS                                  | Specifies how to draw tic marks on lower edge.                                    |
| CURVELINESYMBOL                             | Selects lines and/or symbols to be drawn through the x-y points.                  |
| XDIVISIONS                                  | Specifies spacing of tic marks on the x-axis for all curves.                      |
| XVALUE PRINT SKIP                           | Specifies how often to print the x-values alongside the x-axis tic marks.         |
| CLEAR                                       | Resets X-Y Plot commands to their default value.                                  |
| XTITLE                                      | Defines a character string that will appear along the x-axis.                     |
| TCURVE                                      | Defines a character string that will appear at the top of the plot frame.         |
| LONG  | Controls amount of curve's summary printout.                                      |
| CSCALE                                      | Defines scale factor for characters in the plot frame.                            |
| Commands Applied to Whole Frame Curves Only |   |
| YMIN  | Specifies the minimum value on the y-axis.  |
| YMAX  | Specifies the maximum value on the y-axis.  |
| XAXIS                                       | Controls the plotting of the x-axis.  |
| YINTERCEPT                                  | Specifies the location of the y-axis on the x-axis.                               |
| YLOG  | Selects logarithmic or linear y-axis.   |
| LEFT TICS                                   | Specifies how to draw tic marks on left edge.                                     |
| RIGHT TICS                                  | Specifies how to draw tic marks on right edge of the frame.                       |
| ALLEDGE TICS                                | Specifies how to draw tic marks on all edges of the frame.                        |
| YDIVISIONS                                  | Specifies spacing of tic marks on the y-axis.                                     |
| YVALUE PRINT SKIP                           | Specifies how often to print the y-values alongside the y-axis tic marks applies. |

| Commands Applied to Whole Frame Curves Only      |  |
|--|--|
| XGRID LINES                                      | Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks. |
| YGRID LINES                                      | Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks. |
| YTITLE   | Defines a character string that will appear along the y-axis                           |
| Commands Applied to Upper Half Frame Curves Only |  |
| YTMIN  | Specifies the minimum value on the y-axis.   |
| YTMAX  | Specifies the maximum value on the y-axis.   |
| YTAXIS   | Controls the plotting of the y-axis.   |
| YTINTERCEPT                                      | Specifies the location of the y-axis on the x-axis.                                    |
| YTLOG  | Selects logarithmic or linear y-axis.  |
| TLEFT TICS                                       | Specifies how to draw tic marks on the left edge.                                      |
| TRIGHT TICS                                      | Specifies how to draw tic marks on the right edges.                                    |
| TALL EDGE TICS                                   | Specifies how to draw tic marks on all edges.  |
| YTDIVISIONS                                      | Specifies spacing of tic marks on the y-axis.  |
| YTVALUE PRINT                                    | Specifies how often to print the y-values alongside the y-axis tic marks.              |
| SKIP   |  |
| XTGRID LINES                                     | Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks. |
| YTGRID LINES                                     | Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks. |
| YTITLE   | Defines a character string that will appear along the y-axis.                          |
| Commands Applied to Lower Half Frame Curves Only |  |
| YBMIN  | Specifies the minimum value on the y-axis.   |
| YBMAX  | Specifies the maximum value on the y-axis.   |
| XBAXIS   | Controls the plotting of the x-axis.   |
| YBINTERCEPT                                      | Specifies the location of the y-axis on the x-axis.                                    |
| YBLOG  | Selects logarithmic or linear y-axis.  |
| BLEFT TICS                                       | Specifies how to draw tic marks on left edge.  |
| BRIGHT TICS                                      | Specifies how to draw tic marks on right edge.   |
| BALL EDGE TICS                                   | Specifies how to draw tic marks on all edges.  |
| YBDIVISIONS                                      | Specifies spacing of tic marks on the y-axis.  |

| Commands Applied to Lower Half Frame Curves Only |  |
|--|--|
| <a href="#">YBVALUE PRINT</a>                    | Specifies how often to print the y-values alongside the y-axis tic marks.              |
| <a href="#">SKIP</a>                             |  |
| <a href="#">XBGRID LINES</a>                     | Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks. |
| <a href="#">YBGRID LINES</a>                     | Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks. |
| <a href="#">YBTITLE</a>                          | Defines a character string that will appear along the y-axis.                          |

| X-Y Plot Generation Commands |  |
|------------------------------|--|
| <a href="#">XYPAPLOT</a>     | Generate X-Y plots for a printer.              |
| <a href="#">XYPEAK</a>       | Print only the summary for all curves.         |
| <a href="#">XYPLOT</a>       | Generate X-Y plots for a plotter.              |
| <a href="#">XYPRINT</a>      | Generate table of X-Y pairs for a printer.     |
| <a href="#">XYPUNCH</a>      | Generate table of X-Y pairs for the PUNCH file |

## ALLEDGE TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the frame.

Format:

ALLEDGE TICS tic

Example:

ALLEDGE -1

| Descriptor | Meaning  |
|------------|--|
| tic        | Specifies how to draw tic marks (Integer, Default= 0). |
| -1         | Draw tic marks only.                                   |
| 0          | Do not draw tic marks or associated values (Default).  |
| 1          | Draw tic marks and associated values.                  |

Remarks:

1. ALLEDGE TICS applies to whole frame curves only.
2. To determine if on any given edge (a) tic marks will be drawn without values, (b) no tic marks or values will be drawn, or (c) tic marks with values will be drawn, the following sum must be computed by the user. Add the tic values of the edge in question to its associated ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS tic values. If the resulting value is less than zero, tic marks will be drawn without values. If the resulting value is zero, no tic marks or values will be drawn. If the resulting value is greater than zero, tic marks with values will be drawn. The user should be careful in the use of the ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS commands. For example, the use of only the ALLEDGE TICS = -1 command will result in no tic marks or values being drawn, since the default values for individual edges is +1. Tic values input may only be -1, 0, or 1.

**BALL EDGE TICS****Controls Drawing of Tic Marks on Lower Half**

Specifies how to draw tic marks on lower half of frame.

**Format:**

BALL EDGE TICS tic

**Example:**

BALL EDGE TICS -1

| Describer | Meaning   |
|-----------|---|
| tic       | Specifies how to draw tic marks (Integer, Default = 0). |
| -1        | Draw tic marks only.                                    |
| 0         | Do not draw tic marks or associated values (Default).   |
| 1         | Draw tic marks and associated values.                   |

**Remarks:**

1. BALL EDGE TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

## BLEFT TICS

Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of lower half of frame.

**Format:**

BLEFT TICS tic

**Example:**

BLEFT TICS -1

| Descriptor | Meaning  |
|------------|--|
| tic        | Specifies how to draw tic marks (Integer; Default= 1). |
| -1         | Draw tic marks only.                                   |
| 0          | Do not draw tic marks or associated values.            |
| 1          | Draw tic marks and associated values (Default).        |

**Remarks:**

1. BLEFT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [BRIGHT TICS, 669](#).

## BRIGHT TICS

### Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of lower half of frame.

**Format:**

BRIGHT TICS tic

**Example:**

BRIGHT TICS -1

| Descriptor | Meaning   |
|------------|---|
| tic        | Specifies how to draw tic marks (Integer; Default = 1). |
| -1         | Draw tic marks only.                                    |
| 0          | Do not draw tic marks or associated values.             |
| 1          | Draw tic marks and associated values (Default).         |

**Remarks:**

1. BRIGHT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

**CAMERA****Plotter Media Selection**

Selects plotter media.

**Format:**

CAMERA ctype

**Example:**

CAMERA 1

| Descriptor | Meaning                                       |
|------------|---|
| ctype      | Camera type (Integer 1, 2, or 3; Default= 2). |
| 1          | Film  |
| 2          | Paper (Default)                               |
| 3          | Both  |

**Remark:**

1. If the CAMERA command is not specified then CAMERA 2 is assumed.

## CLEAR Resets X-Y Plot Commands

Resets X-Y Plot commands to their default values.

Format

CLEAR

Remark:

1. All commands except XTITLE, YTITLE, YTTITLE, YBTITLE, and TCURVE will revert to their default values.

## CSCALE

### Character Scale Factor

Defines scale factor for characters in the plot frame. See the command [CSCALE, 750](#) in the OUTPUT(PLOT) Section.

**CURVELINESYMBOL****Curve, Line and Symbol Selection**

Selects lines and/or symbols to be drawn through the x-y points.

**Format:**

**CURVELINESYMBOL symtype**

**Example:**

**CURV 4**

| Descriptor | Meaning  |
|------------|--|
| symtype    | Specifies the symbol drawn at the x-y points. If symtype is 0 then only lines will be drawn through the points with no symbol. If symtype is less than zero then only the symbol and not the lines will be drawn. If symtype is greater than zero then both the symbol and the lines will be drawn (-9 ≤ Integer ≤ 9; Default= 0). |

| symtype | Symbol |
|---------|--------|
| 0       | none   |
| 1       | X      |
| 2       | *      |
| 3       | +      |
| 4       | -      |
| 5       | .      |
| 6       | ×      |
| 7       | [ ]    |
| 8       | <>     |
| 9       | / \    |

**Remark:**

1. If more than one curve is plotted per frame, then the symbol number is incremented by 1 for each curve.

## DENSITY

### Microfilm Plotter Line Density

Selects the line density for microfilm plotters only.

#### Format

DENSITY d

#### Example

DENS 3

| Descriptor | Meaning   |
|------------|---|
| d          | Specifies line density scale factor for microfilm plotters. A line density of d is d times heavier than a line density of 1 (Integer $\geq 0$ ; Default = 1). |

**LEFT TICS****Controls Drawing of Tic Marks on Left Edge**

Specifies how to draw tic marks on left edge of whole frame curves.

**Format:**

LEFT TICS tic

**Example:**

LEFT -1

| Descriptor | Meaning   |
|------------|---|
| tic        | Specifies how to draw tic marks (Integer; Default = 1). |
| -1         | Draw tic marks only.                                    |
| 0          | Do not draw tic marks or associated values.             |
| 1          | Draw tic marks and associated values (Default).         |

**Remarks:**

1. LEFT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [RIGHT TICS, 680](#).

## LONG

### Summary Print Control

Controls amount of curve's summary printout.

Format:

LONG{ YES }  
NO }

| Describers | Meaning                                      |
|------------|--|
| YES        | One page for each curve's summary (Default). |
| NO         | Condensed curve summary.                     |

Remark:

1. If LONG is not specified, then LONG=NO is assumed.

## LOWER TICS

### Controls Drawing of Tic Marks on Lower Edge

Specifies how to draw tic marks on lower edge.

**Format:**

LOWER TICS tic

**Example:**

LOWER -1

| Describers | Meaning  |
|------------|--|
| tic        | Specifies how to draw tic marks (Integer; Default= 1). |
| -1         | Draw tic marks only.                                   |
| 0          | Do not draw tic marks or associated values.            |
| 1          | Draw tic marks and associated values (Default).        |

**Remarks:**

1. LOWER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

## PENSIZE

### Pen Selection

Selects pen number.

**Format:**

PENSIZE p

**Example:**

PENS 3

| Descriptor | Meaning  |
|------------|--|
| p          | Specifies pen number to be used to generate the plot (Integer > 0; Default = 1). |

## PLOTTER

### X-Y Plot File Format

See the command [PLOTTER, 769](#) in the OUTPUT(PLOT) Section.

**RIGHT TICS**

Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of the frame.

**Format:**

RIGHT TICS tic

**Example:**

RIGHT -1

| Describers | Meaning   |
|------------|---|
| tic        | Specifies how to draw tic marks (Integer; Default = 1). |
| -1         | Draw tic marks only.                                    |
| 0          | Do not draw tic marks or associated values.             |
| 1          | Draw tic marks and associated values (Default).         |

**Remarks:**

1. RIGHT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [LEFT TICS, 675](#).

**SEPLOT****Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

**Format:**

SEPLOT seid1 [ seid2 ... ]

**Examples:**

SEPLOT 5

SEPLOT 0 3 7 200

| Descriptor | Meaning   |
|------------|---|
| seidi      | Superelement identification number (Integer $\geq 0$ ). |

**Remarks:**

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTS or XYPLOTS refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

**SEUPPLOT****Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPILOT commands to a superelement and all of its upstream superelements.

**Format:**

SEUPPLOT seid

**Example:**

SEUPPLOT 7

| Descriptor | Meaning   |
|------------|---|
| seid       | Superelement identification number (Integer $\geq 0$ ). |

**Remarks:**

1. See also related command SEPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

## TALL EDGE TICS

## Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the upper half of the frame.

**Format:**

TALL EDGE TICS tic

**Example:**

TALL -1

| Describers | Meaning   |
|------------|---|
| tic        | Specifies how to draw tic marks (Integer; Default = 0). |
| -1         | Draw tic marks only.                                    |
| 0          | Do not draw tic marks or associated values (Default).   |
| 1          | Draw tic marks and associated values.                   |

**Remarks:**

1. TALL EDGE TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

## TCURVE

### Curve Title

Defines a character string that will appear at the top of the plot frame.

**Format:**

TCURVE ctitle

**Example:**

TCUR RIGHT WING -- LOAD CASE 3

| Descriptor | Meaning  |
|------------|--|
| ctitle     | Any character string (Character; Default = blank). |

**Remark:**

1. TCURVE may not be continued to the next command line.

**TLEFT TICS**

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on the left edge of the upper half of the frame.

**Format:**

TLEFT TICS tic

**Example:**

TLEFT -1

| Describers | Meaning  |
|------------|--|
| tic        | Specifies how to draw tic marks (Integer; Default= 1). |
| -1         | Draw tic marks only.                                   |
| 0          | Do not draw tic marks or associated values.            |
| 1          | Draw tic marks and associated values (Default).        |

**Remarks:**

1. TLEFT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [TRIGHT TICS, 686](#).

**TRIGHT TICS**

Controls Drawing of Tic Marks on the Right Edge

Specifies how to draw tic marks on all edges of the upper half of the frame.

**Format:**

TRIGHT TICS tic

**Example:**

TRIGHT -1

| Describers | Meaning   |
|------------|---|
| tic        | Specifies how to draw tic marks (Integer; Default = 1). |
| -1         | Draw tic marks only.                                    |
| 0          | Do not draw tic marks or associated values.             |
| 1          | Draw tic marks and associated values (Default).         |

**Remarks:**

1. TRIGHT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [TLEFT TICS, 685](#).

## UPPER TICS

## Controls Drawing Of Tic Marks On Upper Edge

Specifies how to draw tic marks on upper edge.

**Format:**

UPPER TICS tic

**Example:**

UPPER -1

| Describers | Meaning  |
|------------|--|
| tic        | Specifies how to draw tic marks (Integer; Default= 1). |
| -1         | Draw tic marks only.                                   |
| 0          | Do not draw tic marks or associated values.            |
| 1          | Draw tic marks and associated values (Default).        |

**Remarks:**

1. UPPER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [LOWER TICS, 677](#).

**XAXIS****X-Axis Plot Control**

Controls the plotting of the x-axis on whole frame curves only.

Format:

XAXIS{ YES }  
NO

| Describers | Meaning                           |
|------------|-----------------------------------|
| YES        | Plot the x-axis.                  |
| NO         | Do not plot the x-axis (Default). |

Remarks:

1. XAXIS applies to whole frame curves only.
2. See related command [YAXIS, 712](#).

**XBAXIS****X-Axis Plot Control**

Controls the plotting of the x-axis on lower half frame curves only.

Format:

XBAXIS{ YES }  
          | NO |

| Describers | Meaning                           |
|------------|-----------------------------------|
| YES        | Plot the x-axis.                  |
| NO         | Do not plot the x-axis (Default). |

Remark:

1. XBAXIS applies to lower half frame curves only.

**XBGRID LINES****Plot X-Axis Grid Lines**

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on lower half frame curves only.

Format:

XBGRID LINES { YES }  
                  { NO }

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the x-axis grid lines.                  |
| NO         | Do not plot the x-axis grid lines (Default). |

Remarks:

1. XBGRID applies to lower half frame curves only.
2. See related command [YBGRID LINES, 715](#).

**XDIVISIONS****Tic Spacing on Y-Axis**

Specifies spacing of tic marks on the x-axis for all curves.

**Format:**

XDIVISIONS xd

**Example:**

XDIV 10

| Descriptor | Meaning  |
|------------|--|
| xd         | Number of spaces between tic marks on x-axis (Integer > 0; Default = 5). |

**Remarks:**

1. XDIVISIONS applies to all curves and to the commands: UPPER TICS, LOWER TICS, and YINTERCEPT.
2. XDIVISIONS is ignored for a logarithmic x-axes.

## XGRID LINES

### Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on whole frame curves only.

Format:

XGRID LINES { YES }  
NO

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the x-axis grid lines.                  |
| NO         | Do not plot the x-axis grid lines (Default). |

Remarks:

1. XGRID applies to whole frame curves only.
2. See related command [YGRID LINES, 723](#).

**XINTERCEPT**

Location of X-Axis on Y-Axis

Specifies the location of the x-axis on the y-axis.

**Format:**

XINTERCEPT xi

**Example:**

XINT 50.

| Describer | Meaning   |
|-----------|---|
| xi        | Location of x-axis on the y-axis (Real; Default = 0.0). |

**XLOG****Logarithmic or Linear X-Axis**

Selects logarithmic or linear x-axis.

Format:

XLOG{ YES }  
NO }

| Describers | Meaning                         |
|------------|---------------------------------|
| YES        | Plot a logarithmic x-axis.      |
| NO         | Plot a linear x-axis (Default). |

Remarks:

1. XLOG applies to all curves.
2. The default value for tic division interval depends on the number of log cycles. The default values for tic divisions are given as follows but will range over whole cycles:

| Number of Cycles | Intermediate Values            |
|------------------|--------------------------------|
| 1, 2             | 2., 3., 4., 5., 6., 7., 8., 9. |
| 3                | 2., 3., 5., 7., 9.,            |
| 4                | 2., 4., 6., 8.,                |
| 5                | 2., 5., 8.                     |
| 6, 7             | 3., 6.                         |
| 8, 9, 10         | 3.                             |

**XMAX****Maximum X-Axis Value**

Specifies the maximum value on the x-axis.

**Format:**

XMAX xmax

**Example:**

XMAX 100.

| Describer | Meaning                             |
|-----------|-------------------------------------|
| xmax      | Maximum value on the x-axis. (Real) |

**Remarks:**

1. If XMAX is not specified, then the maximum value is set to the highest value of x.
2. See related commands [XMIN, 696](#), [YMIN, 726](#), and [YMAX, 725](#).

**XMIN**      Minimum X-Axis Value

Specifies the minimum value on the x-axis.

**Format:**

XMIN xmin

**Example:**

XMIN 100.

| Descriptor | Meaning                             |
|------------|-------------------------------------|
| xmin       | Minimum value on the x-axis (Real). |

**Remarks:**

1. XMIN applies to all curves.
2. If XMIN is not specified, then the minimum value is set to the lowest value of x.
3. See related commands [XMAX, 695](#), [YMIN, 726](#), and [YMAX, 725](#).

**XPAPER****Paper Size in X-Direction**

Defines the size of the paper in x-direction.

**Format:**

XPAPER xszie

**Example:**

XPAP 10.

| Describer | Meaning  |
|-----------|--|
| xsize     | Size of paper in x-direction and in inches (Real; Default = 20.0). |

**Remarks:**

1. The default paper size is 20 by 20 inches.
2. See related command [YPAPER, 727](#).

## XTAXIS

### X-Axis Plot Control

XYAXIS applies to upper half frame curves only.

Format:

XTAXIS{ YES }  
NO

| Describers | Meaning                           |
|------------|-----------------------------------|
| YES        | Plot the x-axis.                  |
| NO         | Do not plot the x-axis (Default). |

## XTGRID LINES

### Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on upper half frame curves only.

Format:

XTGRID LINE { YES }  
                  { NO }

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the x-axis grid lines.                  |
| NO         | Do not plot the x-axis grid lines (Default). |

Remarks:

1. XTGRID applies to upper half frame curves only.
2. See related command [YTGRID LINES, 730](#)

## XTITLE

X-Axis Title

Defines a character string that will appear along the x-axis.

**Format:**

XTITLE xtit

**Example:**

XTIT RIGHT WING CASE 3 - TIME

| Descriptor | Meaning  |
|------------|--|
| xtit       | Any character string (Character; Default = Blank). |

**Remarks:**

1. XTITLE may not be continued to the next command line.
2. XTITLE applies to all curves.

## XYPAPLOT

### Generate X-Y Plots for a Printer

Generates X-Y plots for a printer.

See [XYPLLOT, 703](#) for format, describers, and additional remarks.

#### Remarks:

1. The x-axis moves vertically along the page and the y-axis moves horizontally along the page.
2. An asterisk (\*) identifies the points associated with the first curve of a frame, and then for successive curves on a frame the points are designated by symbols O, A, B, C, D, E, F, G, and H.

## XYPEAK

### Print Curve Summary

Print only the summary for all curves. The summary output is titled:

"X Y - O U T P U T S U M M A R Y"

and is also printed under XYPILOT, XYPUNCH, XYPRINT, and XYPAPLOT. This output contains the maximum and minimum values of y for the range of x.

See [XYPILOT, 703](#) for format, describers, and additional remarks.

**XYPLOT****Generate X-Y Plots for a Plotter**

Generates X-Y Plots for a plotter.

**Format:**

```
XYPLOT      yvtype ptype [ i1, i2, i3,... ] /
             id11 (itemu11 [, iteml11] ), id12 (itemu12 [, iteml12] ) , ... /
             id21 (itemu21 [, iteml21] ), id22 (itemu22 [, iteml22] ) , ... / ....
```

**Examples:**

BEGIN BULK or OUTPUT(PLOT) commands are shown as a reminder to place X-Y output request packets properly in the Case Control Section; i.e., at the end of the Case Control Section, or just ahead of any structure plot requests.

**Example 1:**

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XYPLOT SDISP/16(T1)
BEGIN BULK
```

This sequence causes a single whole frame to be plotted for the T1 displacement component of solution set point 16 using the default parameter values. If 16(T1) is not in the solution set, a warning message will be printed and no plot will be made. The plot will be generated for the NASTRAN plotter on file PLT, which must be available.

**Example 2:**

```
OUTPUT (XYOUT)
CSCALE = 1.8
XYPLOT, XYPRINT VELO RESPONSE 1,5 /3(R1,), 5(,R1)
OUTPUT (PLOT)
```

This sequence causes two frame plots (each consisting of an upper half frame and a lower half frame) to be plotted, one for subcase 1 and one for subcase 5, using the default parameter values. The velocity of the first rotational component of grid point 3 will be plotted on the upper half frame, and that of grid point 5 will be plotted on the lower half frame. Tabular printer output will also be generated for both curves.

**Example 3:**

```
OUTPUT (XYPLOT)
CSCALE = 1.8
YDIVISIONS = 20
XDIVISIONS = 10
SGRID LINES = YES
YGRID LINES = YES
XYPLOT DISP 2,5/10(T1),10(T3)
BEGIN BULK
```

This sequence causes two whole frame plots to be generated, one for subcase 2 and one for subcase 5. Each plot contains the T1 and T3 displacement component for grid point 10. The default parameters will be modified to include grid lines in both the x-direction and y-direction, with 10 spaces in the x-direction and 20 spaces in the y-direction. The plot will be generated for the NASTRAN plotter on file .plt.

#### Example 4

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XAXIS = YES
YAXIS = YES
XPAPER = 40.
YPAPER = 20.
XYPLOT STRESS 3/ 15(2) / 21(7)
OUTPUT (PLOT)
```

This sequence causes two whole frame plots to be generated using the results from subcase 3. The first plot is the response of the axial stress for rod element number 15. The second plot is the response of the major principal stress at Z1 for CTRIA3 element number 21. The default parameters will be modified to include the x-axis and y-axis drawn through the origin. Each plot will be initially scaled to fit on 40 x 20 inch paper. The plots will be generated for the NASPLT postprocessor and NASTRAN file .plt2 which must be defined. NASPLT will redefine the plot to 14 x 7-inch paper (with default options).

#### Example 5

```
OUTPUT (XYPLOT)
CURVELINESYMBOL = -1
XYPLOT XYPA PLOT VG / 1(G,F) 2(G,F) 3(G,F) 4(G,F)
OUTPUT (PLOT)
```

This sequence is an example of plotting in a flutter analysis for which a split frame plot is made; the upper half is V-g and the lower half is V-f. Data from the first four loops will be plotted. Distinct symbols are used for data from each loop, and no lines are drawn between points (since the flutter analyst must sometimes exercise judgement about which points should be connected). The plots will also be printed in the normal output. These plots will not have all the features of the external plots, but can be very useful in getting a quick picture of the curves.

#### Example 5

```
XTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ SC 200
YTITLE=FLUID MODE PF AT FLUID POINT 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) MODE 200/204
```

#### Example 6

```
YTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ
YTITLE=PSD MPF FOR FLUID GRID 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) PSDF /204
```

#### Example 7

```
XYPLOT CSIG PSDF / 4 (2003)
```

This is an example of plotting Composite Stress value of stress quantity given by Item Code 3 for layer with ID 2 of element with ID 4.

This can also be equivalently written as (see Remark 9 for explanation)

`XYPLOT CSIG PSDF / 4 (-2,3)`

| Describers  | Meaning  |
|---|--|
| yvtype  | Type of y-value to be plotted: (Character).          |
| ACCE  | Acceleration in the physical set.                    |
| BOUT  | Slideline contact output.                            |
| DISP  | Displacement in the physical set.                    |
| ELFORCE   | Element force.                                       |
| ENTHALPY  | Enthalpy in the physical set.                        |
| FLUX  | Element heat flux.                                   |
| FMPF<br>(mode_id or<br>frequency_id)                                  | Fluid mode participation factors. See Remark 8.      |
| GMPF<br>(mode_id or<br>frequency_id,<br>panel_name,<br>panel_grid_id) | Panel grid mode participation factors. See Remark 8. |
| HDOT  | Rate of change of enthalpy in the physical set.      |
| MPCF  | Multipoint force of constraint.                      |
| LMPF  | Load mode participation factors. See Remark 8.       |
| NONLINEAR   | Nonlinear applied load.                              |
| OLOAD   | Applied load.  |
| PMPF<br>(mode_id or<br>frequency_id,<br>panel_name)                   | Panel mode participation factors. See Remark 8.      |
| PRESSURE  | Pressure of fluid-structure body.                    |
| SACCE   | Acceleration in the solution set.                    |
| SDISP   | Displacement in the solution set.                    |
| SMPF<br>(mode_id or<br>frequency_id)                                  | Structural mode participation factors. See Remark 8. |
| SPCF  | Single-point force of constraint.                    |
| STEMP   | Temperature in the solution set.                     |

| Describers | Meaning   |
|------------|---|
| STRAIN     | Element strain.   |
| STRESS     | Element stress.   |
| SVELO      | Velocity in the solution set.   |
| TEMP       | Temperature in the physical set.  |
| VECTOR     | Displacement in the physical set.   |
| VELO       | Velocity in the physical set.   |
| VG         | Flutter analysis.   |
| CSIG       | Composite Stress  |
| CEPS       | Composite Strain  |
| CFAI       | Composite Failure Index   |
| CSRS       | Composite Strength Ratio  |
| ptype      | Plot type defining the meaning of i1, i2, ... etc., idi, itemui, and itemli (Character, Default=“RESPONSE”).  |
| AUTO       | Autocorrelation function on whole frame curves only.  |
| FREQ       | Frequency—for given excitation frequency plot mode participation versus natural frequency—Oxx2E tables—point plot only. See Remark 8.   |
| MODE       | Mode - for given fluid mode plot mode participation versus excitation frequency - Oxx2M tables.   |
| PSDF       | Power spectral density function on whole frame curves only.   |
| RESPONSE   | Time or frequency in SORT2 format, or grid point identification numbers in SORT1 format (Default) .   |
| SPECTRAL   | Response spectrum on whole frame curves only.   |
| i1, i2,... | Subcase identification numbers for ptype=RESPONSE. The list must be specified in ascending order. For ptype=SPECTRAL, the subcase refers to the RECNO in the DTI,SPSEL Bulk Data entry. The list is ignored for ptype=AUTO and PSDF (Integer $\geq 0$ , Default is all subcases). |

| Describers           | Meaning   |
|----------------------|---|
| idij                 | Element, grid, scalar, or extra point identification number for y-value for frame i. For yvtype=VG, idij refers to the loop count of a flutter analysis (Integer > 0).  |
| itemuij,<br>itemlij, | Item code for y-value. itemuij is for upper half or whole itemlij curves on frame i, and itemlij is for lower half curves only on frame i. If itemlij is not specified, then whole frame curves will be plotted with itemuij. itemlij is ignored for ptype=“AUTO”, “PSDF”, and “SPECTRAL” (Character or Integer > 0).   |
|                      | For elements, the code represents a component of the element stress, strain, or force and is described in <a href="#">Table 7-1</a> and <a href="#">Table 7-6</a> of the Guide. For ptype=“AUTO” and “PSDF”, the complex stress or strain item codes need to be used. Since the output quantities are real, you can use either the real or the imaginary item code. Both will give the same result. |

For grid points and pty=“RESPONSE”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3, T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, or R3IP, where Ti stands for the i-th translational component, Ri stands for the i-th rotational component, RM means real or magnitude, and IP means imaginary or phase. For scalar or extra points, or heat transfer analysis, use T1, T1RM, or T1IP. The output format of results in PCH file is dictated by the case control request for DISP, STRESS. For example for SOL111:

```
DISP (PLOT, phase, SORT2) = 1
OUTPUT (XYOUT)
XYPUNCH DISP RESP /3 (T3RM)
```

This will punch results in PCH file in Mag/phase format. If the DISP command is missing results will shift to Real/Imaginary the default format for SOL111 displacement.

For grid points and ptype=“AUTO” or “PSDF”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3. For scalar or extra points, use T1.

For yvtype=VG, itemui and/or itemli can be “F” for frequency or “G” for damping.

### Remarks:

1. Multiple XYPILOT, XYPUNCH, XYPRINT, XYPEAK, and/or XYPAPILOT commands may be specified in the OUTPUT(XYPILOT) section.
2. Solution set requests are more efficient, because the time-consuming recovery of the dependent displacements can be avoided.
3. The item codes also appear in printed summaries as “CURVE ID” for grid points as well as element data.

4. The information after each slash ( / ) specifies the curve(s) that are to be plotted on the same frame. The describer idij identifies the grid point j or element j associated with the frame number i. All plot requests on one command are sorted on idij to improve the efficiency of the plotting process. Symbols are assigned in order by idij.
5. If any of the item codes, itemlij or itemuij, are not specified; e.g., (8) or (5), the corresponding half frame curve is not plotted. If both the comma ( , ) and itemlij not specified; e.g., (8), then whole frame curves will be plotted. Also, for any single frame, the specifications of “(itemuij,itemlij)” must be consistently half frame (upper and/or lower) or whole frame. For example on half frame curves, if iteml11 and the comma is not specified then either iteml12 or itemu12 must not be specified and on whole frame curves, the commas, iteml11, and iteml12 must not be specified. In other words, the curves on each plot frame must be all whole or half (upper and/or lower).
6. The XYPLOT command may be continued on the next line as long as “XYPLOT yvtype ptype [ i1, i2, i3,... ] /” is specified on the first line.
7. Specifying a nonexistent grid point may cause the program to exit in the XYTRAN module and missing plots to occur.
8. mode\_id is used for natural frequency selection of Oxx2m participation versus excitation frequency output.  
frequency\_id is used for excitation frequency selection of Oxx2E participation versus natural frequency output. frequency\_id is an integer value; e.g., (2) would represent the second frequency calculated.
9. For yvtype = CSIG, CEPS, CFAI and CSRS, we can have (itemuij, itemlij) or (itemuij) formats. For the (itemuij, itemlij) format, itemuij will be a negative integer and its absolute value indicates the Ply ID; itemlij indicates the composite stress item code. For the (itemuij) format, itemuij will be an integer I = PLYID\*1000 + ITEM\_CODE.
10. Printout in the f06 file coming from X-Y plot commands is not supported by the POST TOCASE command.

**XYPRINT****Generate Table of X-Y Pairs for a Printer**

Generates tabular printer output of the X-Y pairs.

See [XYPLOT, 703](#) for format, describers, and additional remarks.

## XYPUNCH

### Generate Table of X-Y Pairs for the PUNCH File

Generates tabular punch output of the X-Y pairs. Same as XYPRINT except the output is written to the PUNCH file.

See [XYPLOT, 703](#) for format, describers, and additional remarks.

**XVALUE PRINT SKIP****Print Values on X-Axis Tic Marks**

Specifies how often to print the x-values alongside the x-axis tic marks.

**Format:**

XVALUE PRINT SKIP xvps

**Example:**

XVAL 5

| Descriptor | Meaning  |
|------------|--|
| xvps       | Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ). |

**Remark:**

1. XVALUE applies to all curves.

## YAXIS

### Y Axis Plot Control

Controls the plotting of the y-axis on all curves.

Format:

YAXIS{ YES }  
NO }

| Describers | Meaning                           |
|------------|-----------------------------------|
| YES        | Plot the y-axis.                  |
| NO         | Do not plot the y-axis (Default). |

**YBDIVISIONS****Tic Spacing on Y-Axis**

Specifies spacing of tic marks on the y-axis for lower half frame curves only.

**Format:**

YBDIVISIONS ybd

**Example:**

YBDI 10

| Descriptor | Meaning  |
|------------|--|
| ybd        | Number of spaces between tic marks on y-axis (Integer > 0; Default = 5). |

**Remarks:**

1. YBDIVISIONS applies to lower half frame curves only.
2. YBDIVISIONS is ignored for a logarithmic y-axis.

## YBINTERCEPT

Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for lower half frame curves only.

Format:

YBINTERCEPT ybi

Example:

YBINT 50

| Descriptor | Meaning   |
|------------|---|
| ybi        | Location of y-axis on the x-axis (Real; Default = 0.0). |

Remark:

1. YBINTERCEPT applies to lower half frame curves only.

## YBGRID LINES

### Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on lower half frame curves only.

Format:

YBGRID LINES { YES }  
                  { NO }

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the y-axis grid lines.                  |
| NO         | Do not plot the y-axis grid lines (Default). |

Remarks:

1. YBGRID applies to lower half frame curves only.
2. See related command [XBGRID LINES, 690](#)

**YBLOG****Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for lower half frame curves only.

Format:

YBLOG{ YES }  
          | NO |

| Describers | Meaning                         |
|------------|---------------------------------|
| YES        | Plot a logarithmic y-axis.      |
| NO         | Plot a linear y-axis (Default). |

Remarks:

1. YBLOG applies to lower half frame curves only.
2. See Remark 2 under [XLOG, 694](#).

**YBMAX****Maximum Y Axis Value**

Specifies the maximum value on the y-axis for lower half frame curves only.

**Format:**

YBMAX ymax

**Example:**

YBMAX 100

| Describer | Meaning                             |
|-----------|-------------------------------------|
| ymax      | Maximum value on the y-axis (Real). |

**Remarks:**

1. YBMAX applies to lower half frame curves only.
2. If YBMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YBMIN, 718](#).

## YBMIN

### Minimum Y Axis Value

Specifies the minimum value on the y-axis for lower half frame curves only.

**Format:**

YBMIN ymin

**Example:**

YBMIN 100

| Descriptor | Meaning                             |
|------------|-------------------------------------|
| ymin       | Minimum value on the y-axis (Real). |

**Remarks:**

1. YBMIN applies to lower half frame curves only.
2. If YBMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YBMAX, 717](#).

**YBTITLE**

Y-Axis Title

Defines a character string that will appear along the y-axis for lower half frame curves only.

**Format:**

YBTITLE ytit

**Example:**

YBTIT RIGHT WING LOADS - CASE 3

| Descriptor | Meaning   |
|------------|---|
| ytit       | Any character string (Character; Default= Blank). |

**Remarks:**

1. YBTITLE may not be continued to the next command line.
2. YBTITLE applies to lower half frame curves only.

## YBVALUE PRINT SKIP

### Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on lower half frame curves only.

**Format:**

YBVALUE PRINT SKIP yvps

**Example:**

YBVAL 5

| Descriptor | Meaning  |
|------------|--|
| yvps       | Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ). |

**Remark:**

1. YBVALUE applies to lower half frame curves only.

**YDIVISIONS****Tic Spacing on Y Axis**

Specifies spacing of tic marks on the y-axis for whole frame curves only.

**Format:**

**YDIVISIONS** *yd*

**Example:**

**YDIV** 10

| Descriptor | Meaning  |
|------------|--|
| <b>yd</b>  | Number of spaces between tic marks on y-axis (Integer > 0; Default = 5). |

**Remarks:**

1. YDIVISIONS applies to whole frame curves only and to the commands: LEFT TICS, RIGHT TICS, and XINTERCEPT.
2. YDIVISIONS is ignored for a logarithmic y-axis.

## YINTERCEPT

Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for whole frame curves only.

Format:

YINTERCEPT yi

Example:

YINT 50

| Descriptor | Meaning   |
|------------|---|
| yi         | Location of y-axis on the x-axis. (Real; Default = 0.0) |

Remark:

1. YINTERCEPT applies to lower half frame curves only.

## YGRID LINES

### Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on whole frame curves only.

Format:

YGRID LINES { YES }  
                  { NO }

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the y-axis grid lines.                  |
| NO         | Do not plot the y-axis grid lines (Default). |

Remarks:

1. YGRID applies to whole frame curves only.
2. See related command [XGRID LINES, 692](#)

**YLOG****Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for whole frame curves only.

Format:

YLOG{ YES }  
NO }

| Describers | Meaning                         |
|------------|---------------------------------|
| YES        | Plot a logarithmic y-axis.      |
| NO         | Plot a linear y-axis (Default). |

Remarks:

1. YLOG applies to whole frame curves only.
2. See Remark 2 under [XLOG, 694](#).

**YMAX****Maximum Y Axis Value**

Specifies the maximum value on the y-axis.

**Format:**

**YMAX ymax**

**Example:**

**YMAX 100**

| Describer | Meaning                             |
|-----------|-------------------------------------|
| ymax      | Maximum value on the y-axis (Real). |

**Remarks:**

1. If YMAX is not specified, then the maximum value is set to the highest value of y.
2. See related command [YMIN, 726](#).

**YMIN**      Minimum Y Axis Value

Specifies the minimum value on the y-axis.

**Format:**

YMIN ymin

**Example:**

YMIN 100

| Descriptor | Meaning                             |
|------------|-------------------------------------|
| ymin       | Minimum value on the y-axis (Real). |

**Remarks:**

1. YMIN applies to all curves.
2. If YMIN is not specified, then the minimum value is set to the lowest value of y.
3. See related command [YMAX, 725](#).

**YPAPER****Paper Size in Y-Direction**

Defines the size of the paper in y-direction.

**Format:**

YPAPER ysize

**Example:**

YPAP 10

| Descriptor | Meaning  |
|------------|--|
| ysize      | Size of paper in y-direction and in inches (Real; Default = 20.0). |

**Remarks:**

1. The default paper size is 20 by 20 inches.
2. See related command [XPAPER, 697](#).

## YTAXIS

### Y Axis Plot Control

Controls the plotting of the y-axis on upper half frame curves only.

Format:

YTAXIS{ YES  
      NO }

| Describers | Meaning                           |
|------------|-----------------------------------|
| YES        | Plot the x-axis.                  |
| NO         | Do not plot the x-axis (Default). |

Remark:

1. YTAXIS applies to upper half frame curves only.

## YTDIVISIONS

### The Spacing on Y Axis

Specifies spacing of tic marks on the y-axis for upper half frame curves only.

**Format:**

YTDIVISIONS ytd

**Example:**

YTDI 10

| Descriptor | Meaning  |
|------------|--|
| ytd        | Number of spaces between tic marks on y-axis (Integer > 0; Default = 5). |

**Remarks:**

1. YTDIVISIONS applies to upper half frame curves only.
2. YTDIVISIONS is ignored for a logarithmic y-axis.

**YTGRID LINES****Plot Y Axis Grid Lines**

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on upper half frame curves only.

Format:

TYGRID LINES { YES }  
                  NO

| Describers | Meaning                                      |
|------------|--|
| YES        | Plot the y-axis grid lines.                  |
| NO         | Do not plot the y-axis grid lines (Default). |

Remarks:

1. YTGRID applies to upper half frame curves only.
2. See related command [XTGRID LINES, 699](#)

**YTINTERCEPT****Location of Y Axis on X Axis**

Specifies the location of the y-axis on the x-axis for upper half frame curves only.

**Format:**

YTINTERCEPT yti

**Example:**

YTINT 50

| Describer | Meaning  |
|-----------|--|
| yti       | Location of y-axis on the x-axis (Real; Default= 0.0). |

**Remark:**

1. YTINTERCEPT applies to upper half frame curves only.

## YTITLE Y Axis Title

Defines a character string that will appear along the y-axis for whole frame curves only.

**Format:**

YTITLE ytit

**Example:**

YTIT RIGHT WING LOADS - CASE 3

| Descriptor | Meaning  |
|------------|--|
| ytit       | Any character string (Character; Default = Blank). |

**Remarks:**

1. YTITLE may not be continued to the next command line.
2. YTITLE applies to whole frame curves only.

**YTLOG****Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for upper half frame curves only.

Format:

YT(LOG){ YES }  
NO }

| Describers | Meaning                         |
|------------|---------------------------------|
| YES        | Plot a logarithmic y-axis.      |
| NO         | Plot a linear y-axis (Default). |

Remarks:

1. YTLOG applies to upper half frame curves only.
2. See Remark 2 under [XLOG, 694](#).

## YTMAX

### Maximum Y Axis Value

Specifies the maximum value on the y-axis for upper half frame curves only.

**Format:**

YTMAX ymax

**Example:**

YTMAX 100

| Describer | Meaning                             |
|-----------|-------------------------------------|
| ymax      | Maximum value on the y-axis (Real). |

**Remarks:**

1. YTMAX applies to upper half frame curves only.
2. If YTMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YTMIN, 735](#).

**YTMIN****Minimum Y Axis Value**

Specifies the minimum value on the y-axis for upper half frame curves only.

**Format:**

YTMIN ymin

**Example:**

YTMIN 100

| Describer | Meaning                             |
|-----------|-------------------------------------|
| ymin      | Minimum value on the y-axis (Real). |

**Remarks:**

1. YTMIN applies to upper half frame curves only.
2. If YTMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YTMAX, 734](#).

## YTTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for upper half frame curves only.

**Format:**

YTTITLE ytit

**Example:**

YTTIT RIGHT WING LOADS - CASE 3

| Descriptor | Meaning  |
|------------|--|
| ytit       | Any character string (Character; Default = Blank). |

**Remarks:**

1. YTTITLE may not be continued to the next command line.
2. YTTITLE applies to upper half frame curves only.

## YTVALUE PRINT SKIP

### Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on upper half frame curves only.

**Format:**

YTVALUE PRINT SKIP yvps

**Example:**

YTVAL 5

| Describer | Meaning  |
|-----------|--|
| yvps      | Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ). |

**Remark:**

1. YTVALUE applies to upper half frame curves only.

## YVALUE PRINT SKIP

Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on whole frame curves only.

**Format:**

YVALUE PRINT SKIP yvps

**Example:**

YVAL 5

| Descriptor | Meaning  |
|------------|--|
| yvps       | Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ). |

**Remark:**

1. YVALUE applies to whole frame curves only.

## OUTPUT(POST) Commands

### SURFACE

#### Surface Definition

Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

```
SURFACE id SET sid, [FIBRE {ALL  
Z1  
Z2  
MID}],  
[SYSTEM {ELEMENT  
BASIC  
CORD cid}], [AXIS {X1  
X2  
X3}], [NORMAL [M]{R  
X1  
X2  
X3}],  
[TOPOLOGICAL  
GEOMETRIC][TOLERANCE{THETA}], [BRANCH [{MESSAGE  
NOMESSAGE}], [BREAK  
NOBREAK]]]
```

Example:

SURFACE 10 SET 9 NORMAL X3

| Descriptor | Meaning  |
|------------|--|
| id         | Surface identification number (Required).  |
| SET        | References a SET command that defines the elements in the surface (required). Either form of the SET command may be used.              |
| sid        | Set identification number.   |
| FIBRE      | Specifies the fiber location at which stresses will be calculated.   |
| ALL        | Requests output at all fiber locations; i.e., z=Z1, Z2, and MID.   |
| Z1         | Requests output z = Z1 only.   |
| Z2         | Requests output z = Z2 only.   |
| MID        | Requests output z = (Z1+Z2)/2 only.  |
| SYSTEM     | Specifies the coordinate system to be used as the output coordinate system.  |
| ELEMENT    | Specifies the element coordinate system for output.  |
| CORD cid   | Specifies the coordinate system defined on a CORDij Bulk Data entry for output.  |
| BASIC      | Specifies the basic coordinate system for output.  |
| AXIS       | Specifies the axis of the coordinate system to be used as the x output axis and the local x-axis when geometric interpolation is used. |

| Descriptor               | Meaning   |
|--------------------------|---|
| X1, X2, X3               | Specifies the direction of the axis or the normal. X, Y, and Z may be substituted for X1, X2, and X3, respectively.   |
| NORMAL                   | Specifies the reference direction for positive fiber and shear stress output, but has no effect when ELEMENT is specified.  |
| M                        | Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter.    |
| R                        | Specifies the radius vector from the origin of reference coordinate system to the grid point.   |
| TOPOLOGICAL<br>GEOMETRIC | Specifies the method to calculate the average grid point stress or strain. The default is TOPOLOGICAL.  |
| theta                    | Specifies the interelement slope difference tolerance (in degrees) to detect stress discontinuity between elements (not used with TOPOLOGICAL) (Real; Default = 0.0). |
| BRANCH                   | Selects whether multiple element intersections (BREAK/NOBREAK) are to be treated as discontinuities, and if warning messages (MESSAGE/NOMESSAGE) are to be issued.    |
| BREAK<br>NOBREAK         | Multiple element intersections are (or are not) to be treated as stress discontinuities.  |
| MESSAGE<br>NOMESSAGE     | A warning message will (or will not) be issued when multiple element intersections are encountered.   |

**Remarks:**

1. SURFACE commands must be specified after OUTPUT(POST).
2. The surface identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands. The sid on the surface entry must reference a SET defined after OUTPUT(POST).
3. The surface normal is also used in the definition of the local reference surface for geometric interpolation. Two options are available. In the first option, the radius vector (R) from the origin of the reference coordinate system to the grid point is used. In the second option, one axis (X1, X2, or X3) of the coordinate system is used. The direction can be reversed using the modification parameter M. The positive side of an element is defined as a side from which the NORMAL direction emerges, rather than the side determined by the connection specified on the element connection entries.
4. When the parameter ELEMENT is present, the element stresses or strains are used unmodified (defaults to output stresses in output system). The CORD keyword references a CORDij Bulk Data entry with coordinate system identification number cid.

5. When theta = 0, no testing is made. When theta is negative, grid point stresses will be calculated for each element connected to an exception point; otherwise, the best estimation of the grid point stress will be output.
6. BREAK is the default if theta is nonzero.
7. For all elements defined in SET 9 of the preceding example,
  - All fiber locations are output;
  - The basic output system is used;
  - The x-axis is x-axis of the basic system;
  - The surface normal direction point is z-axis of the basic system;
  - The topological interpolation method is used;
  - No tolerance test is made; and
  - No branch test is made.

The example illustrates a good choice for regular two-dimensional problems in the x-y plane.

**VOLUME****Volume Definition**

Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

**Format:**

VOLUME id SET sid, [PRINCIPAL, DIRECT STRESS], **SYSTEM** { ELEMENT  
CORD cid  
**BASIC** }

**Example:**

VOLUME 21 SET 2

| Descriptor | Meaning  |
|------------|--|
| id         | Volume identification number.  |
| sid        | Set identification number of a SET command that defines the elements in the volume. Either form of the SET command may be used. The default is all elements.   |
| PRINCIPAL  | Requests principal stresses or strains, direction cosines, mean pressure, and von Mises equivalent stresses or strains to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both. |
| DIRECT     | Requests direct stress or strains, mean pressure stress, and von Mises equivalent stress to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.                               |
| SYSTEM     | Used to specify the reference coordinate system used to define the output stress orientation coordinate system.  |
| ELEMENT    | Specifies the element coordinate system.   |
| CORD cid   | Specifies the coordinate system defined on a CORDij entry.   |
| BASIC      | Specifies the basic coordinate system.   |

**Remarks:**

1. VOLUME commands must be specified after OUTPUT(POST).
2. The volume identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON Case Control commands.
3. If ELEMENT is specified, element stresses or strains are not transformed.
4. In the preceding example, for all elements in SET 2:
  - Both PRINCIPAL and DIRECT stress are output.
  - The BASIC output system is used.

## OUTPUT(PLOT) Commands

The PLOT command requests the generation of undeformed, deformed, or contour plots. All other commands specify how the model will be plotted, type of projection, view angles, scales, etc. All commands have default actions if not specified by the user. The FIND command may be used to calculate an optimal SCALE, ORIGIN, and/or VANTAGE POINT to allow the construction of a plot in a user-specified region of the paper or film. All the commands used in the generation of the various plots will be printed out as part of the output, whether they are directly specified, defaulted or established using the FIND command. Initialization of commands to default values occurs only once. Subsequently, these values remain until altered by direct command input. The only exceptions are the view angles, scale factors, vantage points, and the origins. Whenever the plotter or the method of projection is changed, the view angles are reset to the default values, unless they are respecified by the user. In addition, the scale factors, vantage points, and the origin must be respecified by the user.

The commands are listed here in a logical sequence; however, they need not be so specified. Any order may be used, but if a command is specified more than once, the value or choice stated last will be used.

|                                    |  |
|------------------------------------|--|
| <a href="#">PLOTTER</a>            | Selects format of plot file for interpretation by plotter postprocessor.   |
| <a href="#">ORTHOGRAPHIC, etc.</a> | Selects orthographic projection.   |
| <a href="#">PERSPECTIVE</a>        | Selects perspective projection.  |
| <a href="#">STEREOSCOPIC</a>       | Selects stereoscopic projection.   |
| <a href="#">AXES</a>               | Assigns axes of the basic coordinate system to the observer's coordinate system.   |
| <a href="#">VIEW</a>               | Defines the angular relationship between the observer's coordinate system ( $r$ , $s$ , and $t$ axes specified on the AXES command) and the basic coordinate system.                       |
| <a href="#">MAXIMUM DEFORM</a>     | Defines the magnification of the maximum displacement.   |
| <a href="#">SCALE</a>              | Defines reduction, as a scale factor, of the model's dimensions so that the model fits on a plot frame.  |
| <a href="#">DISTORTION</a>         | Specifies the distortion scale factors of axes in the basic coordinate system.   |
| <a href="#">CSCALE</a>             | Defines scale factor for characters in the plot frame.   |
| <a href="#">ORIGIN</a>             | Defines the origin of the plot frame with respect to the origin of the ( $r$ , $s$ , $t$ ) coordinate system defined on the AXES command.  |
| <a href="#">VANTAGE POINT</a>      | Defines the location of the observer with respect to the model in the ( $r$ , $s$ , $t$ ) coordinate system defined on the AXES command for perspective and stereoscopic projections only. |
| <a href="#">PROJECTION</a>         | Defines the separation, along the $r$ -axis, between the observer and the projection plane if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.   |
| <a href="#">OCULAR SEPARATION</a>  | Defines the separation of the left and right eye vantage points along the $s$ -axis for stereoscopic projections.  |
| <a href="#">CAMERA</a>             | Specifies microfilm plotter options.   |

|            |  |
|------------|--|
| PAPER SIZE | Defines the size and type of the paper.  |
| PEN        | Generates a message on the printed output to inform the plotter operator as to what size and color pen point to mount in the various pen holders.                            |
| PTITLE     | Defines a character string that will appear at the top of the plot frame on the line below the sequence number.  |
| SET        | Defines a set of elements and/or grid points to be plotted.  |
| FIND       | Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.                                 |
| CONTOUR    | Specifies contour plot options for stress, displacement, or temperature.   |
| PLOT       | Generates an undeformed plot or a deformed plot per subcase, mode number, frequency, or time step. A contour plot may also be requested with an undeformed or deformed plot. |

**AXES****Orientation of Observer's Coordinate System**

Assigns axes of the basic coordinate system to the observer's coordinate system.

**Format:**

AXES r s t [ SYMMETRIC  
ANTISYMMETRIC ]

**Example 1:** View toward negative x-axis of model.

AXES MX, Y, MZ

**Example 2:** Mirror image of model.

AXES Y X Z

| Describers    | Meaning   |
|---------------|---|
| r,s,t         | Assigns axes of basic coordinate system to axes of observer's coordinate system (Default=X, Y, Z).<br><br>X      X-direction of basic coordinate system (Default for r).<br>Y      Y-direction of basic coordinate system (Default for s).<br>Z      Z-direction of basic coordinate system (Default for t).<br><br>MX     Negative X-direction of basic coordinate system.<br>MY     Negative Y-direction of basic coordinate system.<br>MZ     Negative Z-direction of basic coordinate system. |
| SYMMETRIC     | Specifies a symmetric orientation of the view. See Remark 5. (Default).   |
| ANTISYMMETRIC | Specifies an antisymmetric orientation of the view. See Remark 5.   |

**Remarks:**

1. If no AXES command is specified, then AXES X, Y, Z is the default.
2. The direction of view is in the negative r-direction; i.e., the projection plane is parallel to the s-t plane.
3. The VIEW command depends on the AXES command specification and defines the angular relationship between observer's coordinate system and the basic coordinate system.
4. The AXES command can be used to preposition the object in 90° increments in such a manner that only rotations less than 90° are required by the VIEW command to obtain the desired orientation. MX, MY, MZ can be used to define left-handed coordinate systems. Note that the default system is right-handed.
5. An undeformed or deformed plot of the symmetric portion of an object can be obtained by reversing the sign of the axis that is normal to the plane of symmetry. In the case of multiple planes of symmetry, the signs of all associated planes should be reversed. The ANTISYMMETRIC option should be specified when a symmetric model is loaded in an unsymmetric manner. This will cause

the deformations to be plotted antisymmetrically with respect to the specified plane or planes. Since the AXES command applies to all parts (SETs) of a single frame, symmetric and antisymmetric combinations cannot be made with this command (see the symmetry option on the [PLOT, 761](#) command in this section).

6. To avoid a mirror image, ensure that the r, s, and t axes obey the right-hand rule.

**CAMERA****Microfilm Plotter Options**

Specifies microfilm plotter options.

Format:

CAMERA [ PAPER  
FILM    BLANK FRAME n  
BOTH ]

Example:

CAMERA FILM

| Describers | Meaning   |
|------------|---|
| FILM       | Requests 35 mm or 16 mm film and positive or negative images. |
| PAPER      | Requests positive prints.                                     |
| BOTH       | Requests positive prints and 35 mm or 16 mm film.             |

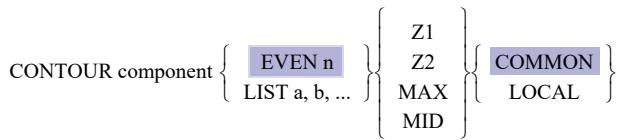
Remarks:

1. If the CAMERA command is not specified, then CAMERA PAPER BLANK FRAMES 0 is assumed.
2. If FILM or BOTH is specified, then these options must be communicated to the plotter operator through normal means of communications at the installation.
3. If FILM or BOTH are specified and if n is greater than 0 then n blank frames will be inserted between plots. The plotter must be operated in the manual mode in order to have blank frames inserted between positive prints. If blank frames are desired only on film, and not on paper, the plotter must be operated in the automatic mode.

**CONTOUR****Contour Plot Options**

Specifies contour plot options for stress, displacement, or temperature.

**Format:**



**Example:**

CONTOUR MAGNIT LIST 2., 4., 6., 8., 10.

| Descriptors    | Meaning   |
|----------------|---|
| component      | Name of stress, displacement or temperature component (Character, Default=“MAJPRIN”).   |
| MAJPRIN        | Major principal stress. Not available for nonlinear elements (Default).   |
| MINPRIN        | Minor principal stress. Not available for nonlinear elements.   |
| EQUIVAL        | von Mises stress. When STRESS(MAXS) is requested in the Case Control Section, von Mises stress is used for plotting. For nonlinear analysis, Mohr-Coulomb or Drucker-Prager stress may also be plotted in conjunction with the MATS1 command. |
| XNORMAL        | X component of normal stress.   |
| YNORMAL        | Y component of normal stress.   |
| ZNORMAL        | Z component of normal stress.   |
| XYSHEAR        | XY component of shear stress.   |
| XZSHEAR        | XZ component of shear stress.   |
| YZSHEAR        | YZ component of shear stress.   |
| XDISP          | T1 component of displacement in global coordinate system.   |
| YDISP          | T2 component of displacement in global coordinate system.   |
| ZDISP          | T3 component of displacement in global coordinate system.   |
| MAGNIT         | Magnitude of displacement or temperature.   |
| EVEN n         | Number of contours ( $50 \geq \text{Integer} > 0$ , Default is EVEN 10).  |
| LIST a, b, ... | List of stresses, displacements or temperatures which define the contours (Real).   |
| Z1             | Stresses at Z1 from neutral plane (Default).  |

| Describers | Meaning  |
|------------|--|
| Z2         | Stresses at Z2 from neutral plane.   |
| MAX        | Maximum of stress at Z1 and Z2.  |
| MID        | Average of stress (membrane stress) at Z1 and Z2.  |
| COMMON     | Plot stress contours in basic coordinate system (Default).   |
| LOCAL      | Plot stress contours in element coordinate system. This is the coordinate system used in printed output. |

**Remarks:**

1. The CONTOUR command should be specified immediately before its associated PLOT command.
2. A STRESS command must appear in the Case Control Section for all elements included in a CONTOUR request. If printed output is not desired, then STRESS(PLOT)=sid should be specified.
3. In linear analysis, stress contour plots are available for the following elements: CTRIA3, CQUAD4, CSHEAR, and CTRIA6. In nonlinear analysis, stress contour plots are available for CQUAD4 and CTRIA3 elements. The Bulk Data element connection entries for all elements must list the grid points in either clockwise or counterclockwise order. Mixing the order will result in meaningless or confusing plots.
4. When selecting contour options, note that
  - MAJPRIN, MINPRIN, EQUIVAL are the same in COMMON and LOCAL.
  - ZNORMAL, XZSHEAR, YZSHEAR, if selected in LOCAL, will be changed to COMMON.
  - CSHEAR elements only have the MAXSHEAR value.
5. The CTRIA6 element stress contour plots are different in that they must be selected as COMMON. Also, the following equivalences apply:

XNORMAL is radial  
 YNORMAL is azimuthal  
 ZNORMAL is axial  
 XYSHEAR is shear  
 XZSHEAR is maximum principal  
 YZSHEAR is von Mises  
 EQUIVAL is octahedra

## CSCALE

### Character Scale Factor

Defines scale factor for characters in the plot frame.

**Format:**

CSCALE cs

**Example:**

CSCA 2.0

| Descriptor | Meaning  |
|------------|--|
| cs         | Scale factor applied to characters in the plot frame (Default = .5). |

**Remarks:**

1. CSCALE is used to control the spacing of characters when plots are made with the NASTRAN plotter and postprocessed with the MSC/NASPLOT routine. For example, if the SCALE FACTOR on the NASPLOT data command is 2.0, a value for cs of 0.5 will result in characters of default size (.07 inches) at the regular spacing. A value of 1.8 produces good spacing when using the postprocessing plotter programs NASTPLT, TEKPLT, and NEUPS. On the other hand, to double the size of both the plot and the characters, the SCALE FACTOR and the CSCALE FACTOR on the NASPLOT data command should both be set equal to 2.0.
2. The CSCALE command must immediately precede the PLOTTER command. If a second CSCALE command is specified, then a second PLOTTER command must also be specified.

## DISTORTION

### Distortion Scale Factors

Specifies the distortion scale factors of the axes in the basic coordinate system.

**Format:**

DISTORTION dx dy dz

**Example:**

DIST 0.5 1.0 1.0

| Describers | Meaning  |
|------------|--|
| dx         | Distortion scale factor of the basic coordinate system's x-axis (Default=1.0). |
| dy         | Distortion scale factor of the basic coordinate system's y-axis (Default=1.0). |
| dz         | Distortion scale factor of the basic coordinate system's z-axis (Default=1.0). |

**Remarks:**

1. If no DISTORTION command is specified, then no distortion is applied.
2. If DISTORTION is specified, then all three values for dx, dy, and dz must be specified even though one or two will use the default.
3. The distortion factors are applied prior to any other scaling commands: SCALE, MAXIMUM DEFORMATION, CSCALE, etc.

**FIND****Automatic Calculation of Scale, Origin, and Vantage Point**

Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.

**Format:**

```
FIND [ SCALE ORIGIN oid VANTAGE POINT SET setid ,
      REGION { le be re te } ]
```

**Example:**

```
FIND SCALE ORIGIN 5 SET 2
```

| Describers | Meaning   |
|------------|---|
| oid        | Origin identification number (Integer>0).   |
| setid      | Set identification number etc. (Integer>0).   |
| le         | Fractional distance of left edge of plot region from the lower left corner of the image area (Real, Default=0.0).   |
| be         | Fractional distance of bottom edge of plot region from the lower left corner of the image area (Real, Default=0.0). |
| re         | Fractional distance of right edge of plot region from the lower left corner of the image area (Real, Default=1.0).  |
| te         | Fractional distance of top edge of plot region from the lower left corner of the image area (Real, Default=1.0).    |

**Remarks:**

1. The FIND command is recommended over the specification of SCALE, ORIGIN, and VANTAGE POINT commands, and should be specified prior to its associated PLOT or CONTOUR command.
2. The FIND command requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN, and/or VANTAGE POINT commands and based on the specification of the
  - PLOTTER command;
  - PROJECTION PLANE command;
  - SET and REGION specifications on the FIND command;
  - VIEW and/or AXES commands;
  - MAXIMUM DEFORMATION command; and
  - PAPER SIZE command. All of these commands must precede the associated FIND command.
3. The FIND command can be used to compute any or all of SCALE, ORIGIN, or VANTAGE POINT as long as they have not been previously specified.
4. If SET is not specified, then the first defined SET will be used.

5. If no options are specified on the FIND command, a SCALE and VANTAGE POINT are selected and an ORIGIN is located, using the first defined SET, so that the plotter object is located within the image area.
6. The plot region is defined as some fraction of the image area (image area = 0.0, 0.0, 1.0, 1.0 and first quadrant = 0.5, 0.5, 1.0, 1.0). The image area is located inside the margins on the paper.

## MAXIMUM DEFORM

## Length of Maximum Displacement

Defines the magnification of the maximum displacement. All other displacements are scaled accordingly.

**Format:**

MAXIMUM DEFORMATION d

**Example:**

Magnify the displacements such that the maximum displacement is equal to two units of length of the model.

MAXI DEFO 2.

| Descriptor | Meaning  |
|------------|--|
| d          | Specifies the length, in units of the model and not of the plot frame, to which the maximum displacement is scaled. (Default=5% of the largest dimension of the model represented by the elements in the SET specification on the PLOT command.) |

**Remarks:**

1. If no MAXIMUM DEFORMATION command is specified, then the previously described default is assumed.
2. If you wish the FIND command to use the d value, a MAXIMUM DEFORMATION command should precede the FIND command.
3. If you wish the plot deformation scaling to be different than the FIND scaling, a different MAXIMUM DEFORMATION command can appear first before the PLOT command.
4. For nonlinear plotting, MAXIMUM DEFORMATION d and the MAXIMUM DEFORMATION field on the PLOT command should have the same value.

## OCULAR SEPARATION

### Separation of the Vantage Points

Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.

**Format:**

OCULAR SEPARATION os

**Example:**

OCULAR SEPARATION 2.0

| Descriptor | Meaning   |
|------------|---|
| os         | Separation, in inches, of the two vantage points along the s-axis (Real, Default=2.756 inches). |

**Remark:**

1. The default value is the separation used in the standard stereoscopic cameras and viewers (70 mm).  
The default value is recommended.

## ORIGIN

### Origin of Plot Frame

Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.

Format:

ORIGIN oid u v

Example:

ORIG 3 -1. -2.

| Descriptors | Meaning  |
|-------------|--|
| oid         | Origin identification number which may be specified after the ORIGIN descriptor on the PLOT command (Integer > 0). |
| u           | Horizontal displacement of plot frame origin from the rst origin (Real, Default = 0.0).                            |
| v           | Vertical displacement of paper origin from the rst origin (Real, Default = 0.0).                                   |

Remarks:

1. In the transformation performed for any of the three projections, the origins of both the basic coordinate system and the observer's coordinate system are coincident. The ORIGIN command may be used to locate the plot frame origin (lower left hand corner) from the rst origin. The units are inches, and are not subject to the scaling of the plotted object.
2. The ORIGIN command is not recommended for general use. See the [FIND, 752](#) command to have the origin optimally located so as to place the plotted object in the center of the plot frame.
3. Ten origins may be specified at one time. However, any one can be redefined at any time. An eleventh origin is also provided if more than ten origins are erroneously defined; i.e., only the last of these surplus origins will be retained.
4. If a projection; e.g., ORTHOGRAPHIC, STEREOSCOPIC, or PERSPECTIVE, is changed in the plot packet, or the PLOTTER command is changed, then all previously defined origins are deleted.

**ORTHOGRAPHIC, etc.****Type of Projection**

Selects type of projection.

Format:

[ORTHOGRAPHIC]  
PERSPECTIVE  
STEREOSCOPIC]

| Describers   | Meaning                                    |
|--------------|--|
| ORTHOGRAPHIC | Selects orthographic projection (Default). |
| PERSPECTIVE  | Selects perspective projection.            |
| STEREOSCOPIC | Selects stereoscopic projection.           |

Remark:

1. If none of the preceding projections are specified, then ORTHOGRAPHIC is used.

## PAPER SIZE

### Paper Dimensions

Defines the size and type of the paper.

**Format:**

PAPER SIZE h X or BY v [ TYPE ptype ]

**Example:**

PAPER SIZE 10. BY 10.

| Describers | Meaning  |
|------------|--|
| h          | Horizontal size of paper in inches (Real, Default = 20.0). |
| v          | Vertical size of paper in inches (Real, Default= 20.0).    |
| ptype      | Paper type (Character, Default = "VELLUM").                |

**Remarks:**

1. The default paper size for the PLOTTER NAST is 20 by 20 inches which is converted to a 7 by 7 inch plot frame by the NASPLOT postprocessor.
2. PAPER SIZE can be specified along with the NASPLOT postprocessor to create rectangular plots. For example, the command will result in a 14 by 7 inch plot frame if the default value of 1.0 is used for the SCALE FACTOR on the NASPLOT command. The SCALE FACTOR on the NASPLOT data command can be used to make larger plots having the shape defined with PAPER SIZE.
3. PAPER SIZE also affects the raster count for the NASTRAN plotter. The default raster count is 1000 for a paper size of 20 by 20. Doubling the paper size to 40 by 40 will double the raster count to 2000.

## PEN      Pen Color and Size Assignments

Generates a message on the printed output which may be used to inform the plotter operator as to what size and color pen point to mount in the various pen holders.

Format:

PEN pn [ COLOR cname ]

Example:

PEN 2 COLOR RED

| Describers | Meaning  |
|------------|--|
| pn         | Pen identification number (Integer > 0).       |
| COLOR      | Flag indicating the next word is a color name. |
| cname      | Pen color (Character).                         |

Remarks:

1. The actual number of pens available will depend on the plotter hardware configuration at each installation.
2. The PEN command does not control the pen used in generating the plot. See the PEN describer on the [PLOT, 761](#) command.
3. The PEN command is optional and is not appropriate for microfilm plotters.

## PERSPECTIVE

Selects Perspective Projection

See the description of the [ORTHOGRAPHIC, etc., 757](#).

**PLOT****Undeformed or Deformed Plot Request**

Generates an undeformed plot of the model or a deformed plot for a subcase, mode number or time step.

**Format:**

```
PLOT [analysis][dtype][CONTOUR][i1, i2 THRU i3, i4, etc.]
      [ { RANGE f1, f2
          TIME t1, t2 } ] ,
```

$$\left[ \begin{array}{l} \text{PHASE LAG } \phi \\ \text{MAGNITUDE} \end{array} \right] [\text{MAXIMUM DEFORMATION } d] ,$$
  

$$[\text{SET sid1}] [\text{ORIGIN oid1}] \left[ \begin{array}{l} \text{SYMMETRY} \\ \text{ANTISYMMETRY} \end{array} \right]_w \left[ \begin{array}{l} \text{PEN} \\ \text{DENSITY} \end{array} \right]_p [\text{SYMBOLS m[,n]]} ,$$
  

$$[\text{LABEL label}] \left[ \begin{array}{l} \text{SHAPE} \\ \text{OUTLINE} \end{array} \right] [\text{VECTOR v}] , [\text{PRINT}] ,$$
  

$$[\text{SHRINK t, o}] [\text{NORMALS}] ,$$
  

$$[\text{SET sid2}] [\text{ORIGIN oid2}] \text{ etc.}$$

**Examples:**

See after Remarks Section.

| Describers | Meaning  |
|------------|--|
| analysis   | Type of analysis results (Character, default results in an undeformed plot or undeformed underlay for contour plots).<br>STATIC      Plot static deformations.<br>MODAL      Plot mode shapes or eigenvectors.<br>CMODAL      Plot complex mode shapes or eigenvectors.<br>TRANSIENT      Plot transient solutions.<br>FREQUENCY      Plot frequency solutions.<br>SENOMOVE      Plot undeformed superelements in their original position; i.e., ignore SELOC Bulk Data entry. |
| dtype      | Specifies plot quantity (Character, Default = "DEFORMATION").<br>DEFORMATI<br>ON      Plot displacements or temperatures in the Z direction (Default).<br>VELOCITY      Plot velocities.<br>ACCELERATI<br>ON      Plot accelerations.  |

| Describers                   | Meaning   |
|------------------------------|---|
| CONTOUR                      | Request for contour plot.   |
| i1, i2,...                   | Subcase identification numbers. See SHAPE and VECTOR for use of "0" (underlay) command. See Remark 3. (Integer $\geq 0$ , Default is all subcases).   |
| RANGE f1 f2                  | Specifies range of natural frequencies, eigenvalues, excitation   |
| TIME t1,t2                   | frequencies, time steps, or load factors. Used to minimize the amount of plotted data. See Remark 4. (Real).  |
| PHASE LAG $\phi$             | Specified phase lag, in degrees, for plotting complex quantities. See Remark 5. (Real, Default = 0.0).  |
| MAGNITUDE                    | Requests magnitude of complex quantities.   |
| MAXIMUM DEFORMATION d        | Specifies the magnification of the maximum displacement. See Remark 6. (Real).  |
| SET sid                      | Set identification number which defines the set of elements or grid points to be plotted (Default is first SET command).  |
| ORIGIN oid                   | Origin identification number (Default is first origin defined by the ORIGIN or FIND command).   |
| SYMMETRY w<br>ANTISYMMETRY w | Request plot of the symmetric portion of the symmetrically or antisymmetrically loaded model. This symmetric portion will be located in the space adjacent to the region originally defined by ORIGIN oid, and will appear as a reflection of the antisymmetrically deformed model about the plane whose normal is oriented parallel to the coordinate direction w. See Remark 7. (Default is no action). |
| PEN p                        | Specifies pen number that is used to generate the plot (Integer $> 0$ ).  |
| DENSITY d                    | Specifies line density scale factor for film plotters which is d times heavier than a line density of 1 (Integer $> 0$ ).   |

| Describers    | Meaning   |
|---------------|---|
| SYMBOLS m[,n] | All the grid points associated with the specified set will have symbol m overprinted with symbol n printed at its location. If n is not specified, only symbol m will be printed. See Remark 8. |

| m or n | Symbol |
|--------|--------|
| 0      | none   |
| 1      | X      |
| 2      | *      |
| 3      | +      |
| 4      | -      |
| 5      | .      |
| 6      | x      |
| 7      | [ ]    |
| 8      | <>     |
| 9      | / \    |

|             |  |
|-------------|--|
| LABEL label | Specifies labeling options at elements and grid points:  |
| GRID        | All the grid points included in the specified set have their identification number printed to the right of the undeformed or deformed location (undeformed location in the case of superimposed plots) (Default).  |
| ELEMENTS    | All the elements included in the specified set are identified by the element identification number and type at the center of each element (undeformed location in the case of superimposed plots). See Remarks 11. and 12.   |
| BOTH        | Both GRID and ELEMENT options.   |
| GSPC        | Label those degrees-of-freedom that are constrained to zero through permanent single point constraints on GRID and GRDSET Bulk Data entries, or are constrained through SPC and SPC1 Bulk Data entries. The label consists of the grid point ID number and the constrained degrees-of-freedom. |
| EPID        | Label elements with their respective property identification (PID) numbers. The label consists of the standard element labels and element PID.   |

| Describers | Meaning  |
|------------|--|
| SHAPE      | All the elements included in the specified set are shown by connecting the associated grid points in a predetermined manner. See Remark 9.   |
| OUTLINE    | Only the outline of all the elements in the specified set is shown. Elements not supported by contour plots are ignored. Outlines are always drawn with PEN 1. See Remark 9.   |
| VECTOR v   | A line will be plotted at the grid points of the set representing by length and direction the deformation of the point. See Remark 10. Possible values of v are:<br><br>X, Y, or Z      Requesting individual components.<br>XY, XZ, or YZ    Requesting two specified components.<br>XYZ                Requesting all three components.<br>RXY, RXZ, or RYZ    Requesting vector sum of two components.<br>RYZ<br>R                  Requesting total vector deformation.<br>N                  Used with any of the preceding combinations to request no underlay shape be drawn. |
| PRINT      | List of the average stresses at the interior grid points in the set will be printed for contour stress plots.  |
| SHRINK t,o | t is the ratio of the reduction to the original dimensions of all two-dimensional elements except the CQUAD8 and CTRIA6 ( $0.0 \leq t \leq 1.0$ ) (Real, Default = 0.1 which results in a 10% reduction).<br><br>o is the ratio of the reduction in length to the original length for one-dimensional elements. There is no default value for o. t must be specified to shrink one-dimensional elements.   |
| NORMALS    | Plot vector normal to CHBDYP and CHBDYG elements.  |

**Remarks:**

1. If PLOT is specified with no describers, then a picture of the undeformed model will be prepared using the first defined set and the first defined origin.
2. Describers analysis through PHASE LAG must be specified in the order shown above.
3. The following should be noted when using subcase numbers for plotting eigenvectors.
  - a. If subcase numbers are specified, then the convention for displacement vectors is that the list of subcases must refer to subcase IDs whenever the number of modes recovered is equal to or less than the number of subcases defined. If the number of modes recovered is more than the subcases defined, the plot request for those modes associated with the subcases must refer to subcase IDs. After the mode associated with the last defined subcase, higher modes will be identified by incrementing the last defined subcase ID by one for each such higher mode.

- b. For the display of element quantities in contour plots, the automatic incrementing beyond the last defined subcase does not occur. All subcase numbers to be plotted must be defined. A MODES command in the Case Control Section may be used for this purpose.
  - c. In problems using cyclic symmetry solution sequences, the plot requests for segments of the model must refer to the coded subcase identification numbers (see [Theory](#) in the *MSC Nastran Reference Guide*). All eigenvectors recovered for the segment will be plotted. The RANGE option can be used to select a subset of all eigenvectors for plotting without use of coded subcase IDs.
  - d. RANGE does not require the use of subcase numbers.
4. RANGE specifies the range of values using requested subcases for which plots will be prepared. If only one number is input, it is used as the lower bound and the upper bound is set to the highest value computed. Unless otherwise noted, the default range is all values computed.
    - a. In real eigenvalue analysis, the values are natural frequencies, in units of cycles per unit time.
    - b. In buckling analysis, the values are eigenvalues.
    - c. In frequency response, the values are excitation frequencies in units of cycles per unit time.
    - d. In transient response, the values are in units of time.
    - e. In static nonlinear analysis (SOLs 106 and 153), the values are load factors. The default range is the highest load factor of each subcase.
    - f. In transient nonlinear analysis (SOLs 129 and 159), the values are in units of time. The default range is the last time step for each subcase.
  5. PHASE LAG  $\phi$  is used in the equation:
$$u_R \cos \phi - u_I \sin \phi$$
where  $u_R$  and  $u_I$  are the real and imaginary parts of the response quantity, respectively. The printed output for magnitude and phase uses the convention of a phase lead.
  6. MAX DEFO is not recommended for general use. Each subcase is separately scaled according to its own maximum if this item is absent. If d is omitted, the set will be scaled to the maximum within the set being plotted.
  7. w specifies the basic coordinates X, Y, or Z, or any combination thereof. This option allows the plotting of symmetric and/or antisymmetric combinations, provided that an origin is selected for the portion of the model defined in the Bulk Data Section that allows sufficient room for the complete plot. This does not permit the combination of symmetric and antisymmetric subcases, as each plot must represent a single subcase. In the case of a double reflection, the figure will appear as one reflected about the plane whose normal is parallel to the first of the coordinates w, followed by a reflection about the plane whose normal is oriented parallel to the second of the coordinates w. This capability is primarily used in the plotting of models that are loaded in a symmetric or an antisymmetric manner. The plane of symmetry must be one of the basic coordinate planes.
  8. Grid points excluded from the set will not have a symbol. Grid points in an undeformed underlay will be identified with symbol 2.

9. In order to get a deformed shape, either SHAPE or OUTLINE must be present in the PLOT command. Both deformed and undeformed shapes or outlines may be specified. All the deformed shapes relating to the subcases listed may be underlaid on each of their plots by including "0" with the subcase string on the PLOT command. The undeformed plot will be drawn using PEN 1 or DENSITY 1 and symbol 2 (if SYMBOLS is specified).
10. All plots requesting the VECTOR option will have an underlay generated of the undeformed shape using the same sets, PEN 1 or DENSITY 1, and symbol 2 (if SYMBOLS is specified). If SHAPE and VECTOR are specified, the underlay will depend on whether "0" is used along with the subcases with DEFORMATION. It will be the deformed shape when not used, and will be both deformed and undeformed shapes when it is used. The part of the vector at the grid point will be the tail when the underlay is undeformed, and the head when it is deformed. If v="N" then no shape will be drawn, but other options such as SYMBOLS will still be valid.
11. Element type labels are: (Plot labels QH and TH indicate hyperelastic elements)

| Element Type | Plot Label |  | Element Type | Plot Label |
|--------------|------------|--|--------------|------------|
| CAERO1       | AE         |  | PLOTEL       | PL         |
| CAXIF2       | A2         |  | CQUAD        | QH         |
| CAXIF3       | A3         |  | CQUAD4       | Q4 or QH   |
| CAXIF4       | A4         |  | CQUAD8       | Q8 or QH   |
| CBAR         | BR         |  | CQUADR       | QR         |
| CBEAM        | BM         |  | CQUADX       | QH         |
| CBEND        | BD         |  | CROD         | RD         |
| CBUSH        | BU         |  | CSHEAR       | SH         |
| CONEAX       | CN         |  | CSLOT3       | S3         |
| CONROD       | CR         |  | CSLOT4       | S4         |
| CDUMI        | Di         |  | CTETRA       | TE         |
| CFLUID2      | F2         |  | CTRIAX6      | D1         |
| CFLUID3      | F3         |  | CTRIA3       | T3 or TH   |
| CFLUID4      | F4         |  | CTRIA6       | T6 or TH   |
| CHBDYG       | HB         |  | CTRIAR       | TR         |
| CHBDYP       | HB         |  | CTRIAX       | TH         |
| CHEXA        | HA         |  | CTUBE        | TU         |
| CPENTA       | HA         |  | CVISC        | VS         |

12. The heat transfer boundary condition elements CHBDYG and CHBDYP can be plotted for undeformed plots. There are several types of CHBDYi elements, as follows:

| Type  | No. of Primary Grid Points | Normals Available |
|-------|----------------------------|-------------------|
| POINT | 1                          | yes               |
| LINE  | 2                          | yes               |
| AREA3 | 3                          | yes               |
| AREA4 | 4                          | yes               |
| REV   | 2                          | no                |
| ELCYL | 2                          | no                |
| TUBE  | 2                          | yes               |
| FTUBE | 2                          | yes               |
| AREA6 | 6                          | yes               |
| AREA6 | 8                          | yes               |

The secondary grid points are used for ambient conditions and are ignored by the plotter. Type POINT must have a nonzero associated area (see AF on the associated PHBDY entry) and a defined normal direction (see V1, V2, V3 on the CHBDYP entry) to be plotted. It is plotted as a hexagon with approximately the correct area. Type LINE must have a nonzero width (see AF on the associated PHBDY entry) and a defined normal in order to plot.

13. To assign PLOT command to superelements it requires an SEUPPLOT or a SEPLOT command.

### Examples:

1. Undeformed SHAPE using first defined SET, first defined ORIGIN and PEN 1 (or DENSITY 1).

PLOT

2. Undeformed SHAPE using SET 3, ORIGIN 4, PEN 2 (or DENSITY 2) with each grid point of the set having a + placed at its location, and its identification number printed adjacent to it.

PLOT SET 3 ORIGIN 4 PEN 2 SHAPE SYMBOLS 3 LABEL

3. Modal deformations as defined in subcase 5 using first defined SET, first defined ORIGIN, and PEN 1 (or DENSITY 1).

PLOT MODAL DEFORMATION 5 SHAPE

4. STATIC deformations as defined in subcases 3, 4, 5, and 8 deformed SHAPE; drawn with PEN 4, using first defined SET and ORIGIN, underlaid with undeformed SHAPE drawn with PEN 1. This command will cause four plots to be generated.

PLOT STATIC DEFORMATION 0, 3 THRU 5, 8 PEN 4, SHAPE

5. Deformations as defined in subcases 1, 2, 3, 4, and 5 undeformed underlay with PEN 1, consisting of SET 2 at ORIGIN 3, SET 2 at ORIGIN 4 (with a < placed at each grid point location), and SET 35 at ORIGIN 4. Deformed data as follows: SHAPE using SET 2 at ORIGIN 3 (PEN 3) and SET 35 at ORIGIN 4 (PEN 4); 3 VECTORS (X, Y and Z) drawn at each grid point of SET 2 at ORIGIN 4 (PEN 4) (less any excluded grid points), with o placed at the end of each vector.

```
PLOT STATIC DEFORMATION 0 THRU 5,
SET 2 ORIGIN 3 PEN 3 SHAPE,
SET 2 ORIGIN 4 PEN 4 VECTORS XYZ SYMBOLS 0,
SET 35 SHAPE
```

6. Static deformations as defined in subcases 3 and 4, both halves of a problem solved by symmetry using the X-Y principal plane as the plane of symmetry. SET 1 at ORIGIN 2 and SET 2 at ORIGIN 3, with the deformed shape plotted using DENSITY 3 and the undeformed model plotted using DENSITY 1. The deformations of the “opposite” half will be plotted to correspond to symmetric loading. This command will cause two plots to be generated.

```
PLOT STATIC DEFORMATIONS 0, 3, 4,
SET 1 ORIGIN 2 DENSITY 3 SHAPE,
SET 1 SYMMETRY Z SHAPE,
SET 2 ORIGIN 3 SHAPE,
SET 2 SYMMETRY Z SHAPE
```

7. Transient deformations as defined in subcase 1 for time = 0.1 to time = 0.2, using SET 1 at ORIGIN 1. The undeformed SHAPE using PEN or DENSITY 1 with an \* at each grid point location will be drawn as an underlay for the resultant deformation vectors using PEN or DENSITY 2 with an < typed at the end of each vector drawn. In addition, a plotted value of 2.0 will be used for the single maximum deformation occurring on any of the plots produced. All other deformations on all other plots will be scaled relative to this single maximum deformation. This command will cause a plot to be generated for each output time step which lies between 0.1 and 0.2.

```
PLOT TRANSIENT DEFORMATION, TIME 0.1, 0.2,
MAXIMUM DEFORMATION 2.0, SET 1, ORIGIN 1, PEN 2, SYMBOLS 2,
VECTOR R
```

8. Contour plot of x-component of normal stress for elements in SET 2 in basic coordinate system at a distance Z1 from neutral plane with 10 contour lines, an outline of elements in SET 2, and using

```
ORIGIN 4.
CONTOUR XNORMAL
PLOT CONTOUR, SET 2, ORIGIN 4, OUTLINE
```

Contour plot of magnitude of displacements at grid points associated with elements in SET 5 with 5 contours having values of 2., 4., 6., 8., 10., and an outline of the elements in SET 5 using

```
ORIGIN 4.
CONTOUR MAGNIT, LIST 2., 4., 6., 8., 10.
PLOT CONTOUR, SET 5, OUTLINE
```

9. Plot the imaginary part of the complex eigenvector in SET 1.

```
PLOT CMODAL DEFORMATION PHASE LAG 90. SET 1 VECTOR R
```

## PLOTTER

### Plot File Format

Selects format of plot file for interpretation by plotter postprocessor.

Format:

PLOTTER { NAST  
          SC }

Example:

PLOTTER NAST

| Describers | Meaning  |
|------------|--|
| NAST       | Specifies format suitable for Postscript plotters (Default). |
| SC         | Specifies Stromberg-Carlson microfilm plotter format.        |

Remark:

1. If no PLOTTER command is specified, then PLOTTER NAST is the default.
2. If “PLOTTER NAST” is specified (or taken by default) and if the format of the PLOT file has been changed to formatted through the use of an ASSIGN statement, plot data will be generated in PostScript mode directly instead of being generated in binary mode. PostScript plot control information may be specified using the SYS= descriptor on the ASSIGN statement or by using the SYSFIELD= command-line keyword, specifying the control information using the PLOT (keyword=value,...) options. The valid keywords are the same as those that can be specified for the PLOTPS or MSCPLOTPS utility programs except that the “begin”, “debug”, “dump”, “end”, “format” and “output” keywords are not allowed.

## PROJECTION

### Separation Between Projection Plane and Observer

Defines the separation along the r-axis, and between the observer and the projection plane, if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.

Format:

PROJECTION PLANE SEPARATION do

Example:

PROJ PLAN SEPA 1.5

| Descriptor | Meaning   |
|------------|---|
| do         | Separation of the observer and the projection plane on the r-axis in model units.<br>The VANTAGE POINT command may also specify the separation (Real, Default = 2.0). |

Remarks:

1. The PROJECTION PLANE SEPARATION command is not recommended. The FIND command is recommended because it automatically calculates the optimum separation.
2. A theoretical description of projection plane separation is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

**PTITLE****Plot Frame Title**

Defines a character string that will appear at the top of the plot frame on the line below the sequence number.

**Format:**

PTITLE ptitle

**Example:**

PTITLE RIGHT WING -- LOAD CASE 3

| Descriptor | Meaning   |
|------------|---|
| ptitle     | Any character string (Character, Default= blank). |

**Remarks:**

1. PTITLE may not be continued to the next command line.
2. Up to four lines of title information will be printed in the lower left-hand corner of each plot. The text for the top three lines is taken from the TITLE, SUBTITLE, and LABEL commands in the Case Control Section. (See the [Case Control Commands, 191](#) for a description of the TITLE, SUBTITLE, and LABEL commands). The text for the bottom line may be of two forms depending on the type plot requested. One form contains the word UNDEFORMED SHAPE. The other form contains the type of plot (statics, modal, etc.) subcase number, load set or mode number, frequency or eigenvalue or time, and (for complex quantities) the phase lag or magnitude. The sequence number for each plot is printed in the upper corners of each frame. The sequence number is determined by the relative position of each PLOT execution command in the plot package. The information on the PTITLE command will be printed on the line below the sequence number. The date and (for deformed plots) the maximum deformation are also printed at the top of each frame.

**SCALE****Scale Factor**

Defines reduction, as a scale factor, of model's dimensions so that model fits on a plot frame.

**Format:**

SCALE a [ b ]

**Example:**

SCALE 0.5

| Describers | Meaning  |
|------------|--|
| a          | Scale factor (Default=1.0).  |
| b          | Ratio of model size/real object size for stereoscopic projection only. |

**Remarks:**

1. The SCALE command is not recommended. The FIND command is recommended because it automatically calculates the optimum scale factor.
2. For orthographic or perspective projections, a is the ratio of the plotted object, in inches, to the real object in the units of model; i.e., one inch of paper equals one unit of model.
3. For stereoscopic projections, the stereoscopic effect is enhanced by first reducing the real object to a smaller model according to b, and then applying a. The ratio of plotted/real object is then the product of a and b.
4. If the NASTRAN general purpose plotter is used in combination with the PLOTPS postprocessing routine, a scale factor can be computed as follows:

$$a = p \cdot \frac{20}{7} \cdot K$$

where:

$p$  = ratio of plot size to object size. For instance, if the model is 100 inches long and the plot size is 7 inches, then

$$p = \frac{7}{100} = .007$$

$\frac{20}{7}$  = ratio of default PAPER SIZE to default PLOTPS frame size.

$K$  = SCALE value on PLOTPS command (Default=1.0). See [Using the Utility Programs](#) in the *MSC Nastran Installation and Operations Guide*.

**SEPLOT****Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPILOT commands to one or more superelements.

**Format:**

SEPLOT seid1 [ seid2 ... ]

**Examples:**

SEPLOT 5

SEPLOT 0 3 7 200

| Descriptor | Meaning   |
|------------|---|
| seidi      | Superelement identification number (Integer $\geq 0$ ). |

**Remarks:**

1. See also related command SEUPPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTS refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

**SET** Set Definition Under OUTPUT(PLOT)

Defines a set of elements or grid point numbers to be plotted.

The SET command specifies sets of elements or grid points, corresponding to portions of the model, which may be referenced by PLOT and FIND commands. The SET command is required. Each set of elements defines, by implication, a set of grid points connected by those elements. The set may be modified by deleting some of its grid points. The elements are used for creating the plot itself and element labeling, while the grid points are used for labeling, symbol printing, and drawing deformation vectors.

Format:

$$\begin{aligned} \text{SET } n = & \left[ \text{ALL} \left[ \begin{array}{l} \text{ELEMENTS} \\ \text{GRID POINTS} \end{array} \right] \left[ \text{EXCEPT} \left\{ \begin{array}{llll} \text{type1} & \text{type2} & \dots & \text{typej} \\ k1 & k2 & \dots & kj \text{ THRU } kk \text{ BY } incj \end{array} \right\} \right] \right], \\ & \left[ \begin{array}{ll} \text{INCLUDE} & \text{ELEMENTS} \\ \text{EXCLUDE} & \text{GRID POINTS} \end{array} \right] \left\{ \begin{array}{llll} \text{type1} & \text{type2} & \dots & \text{typej} \\ k1 & k2 & \dots & kj \text{ THRU } kk \text{ BY } incj \end{array} \right\}, \\ & \left[ \text{EXCEPT} \left\{ \begin{array}{llll} \text{type1} & \text{typem} & \dots & \text{typen} \\ k1 & km & \dots & kn \text{ THRU } ko \text{ BY } incn \end{array} \right\} \right] \end{aligned}$$

Examples:

1. SET 1 consists of elements 1, 5, 10, 11, 13, 14, 15, 20, 22, 24, and 26.

```
SET 1=INCLUDE 1, 5, 10 THRU 15 EXCEPT 12, INCLUDE 20 THRU 26 BY 2
```

2. SET 2 consists of all CTRIA3 and CQUAD4 elements except element 21.

```
SET 2=QUAD4 TRIA3 EXCEPT 21
```

3. SET 10 includes all CTRIAR elements plus elements 70 through 80.

```
SET 10 TRIAR INCLUDE ELEMENTS 70 THRU 80
```

4. SET 15 includes all elements from 15 to 20 and 26 to 100.

```
SET 15=15 THRU 100 EXCEPT 21 THRU 25
```

5. SET 2 includes all elements except CTETRA elements.

```
SET 2=ALL EXCEPT TETRA
```

| Descriptor  | Meaning   |
|-------------|---|
| n           | Sets identification number (0<Integer<999999).                  |
| ALL         | Selects all elements or grid points. See typei.                 |
| ELEMENTS    | Specifies that all identification numbers refer to elements.    |
| GRID POINTS | Specifies that all identification numbers refer to grid points. |

| Describer | Meaning  |
|-----------|--|
| INCLUDE   | Includes specified element or grid point identification numbers or elements in the set.      |
| EXCLUDE   | Excludes specified element or grid point identification numbers or element types in the set. |
| EXCEPT    | Modifies a prior ALL, INCLUDE, or EXCLUDE specification.                                     |
| typei     | Element types. The allowed element types are (Character):                                    |

| Element Type | typei on SET Command |  | Element Type | typei on SET Command |
|--------------|----------------------|--|--------------|----------------------|
| CAXIF2       | AXIF2                |  | CQUAD        | QUAD                 |
| CAXIF3       | AXIF3                |  | CQUAD4       | QUAD4                |
| CAXIF4       | AXIF4                |  | CQUAD8       | QUAD8                |
| CBAR         | BAR                  |  | CQUADR       | QUADR                |
| CBEAM        | BEAM                 |  | CQUADX       | QUADX                |
| CBEND        | BEND                 |  | CROD         | ROD                  |
| CBUSH        | BUSH                 |  | CSHEAR       | SHEAR                |
| CONEAX       | CONE                 |  | CSLOT3       | SLOT3                |
| CONROD       | CONROD               |  | CSLOT4       | SLOT4                |
| CDUMi        | DUMi                 |  | CTETRA       | TETRA                |
| CFLUID2      | FLUID2               |  | CTRIAX6      | TRIAX6               |
| CFLUID3      | FLUID3               |  | CTRIA3       | TRIA3                |
| CFLUID4      | FLUID4               |  | CTRIA6       | TRIA6                |
| CHBDYG       | HBDY                 |  | CTRIAR       | TRIAR                |
| CHBDYP       | HBDY                 |  | CTRIAX       | TRIAX                |
| CHEXA        | HEXA                 |  | CTUBE        | TUBE                 |
| CPENTA       | PENTA                |  | CVISC        | VISC                 |
| PLOTEL       | PLOTEL               |  |              |                      |

|      |  |
|------|--|
| THRU | Specifies a range of identification numbers.     |
| BY   | Specifies an increment for a THRU specification. |
| inci | Increment for THRU range (Integer > 0).          |

**Remarks:**

1. This form of the SET command can only be specified after an OUTPUT(PLOT) delimiter.
2. The INCLUDE, EXCLUDE, and EXCEPT specifications may be specified more than once in the same set. See previous examples.
3. Commas or spaces may be used as separators.
4. Not all of the identification numbers in a THRU range have to correspond to elements or grid points. For example, elements 2, 4, 7, and 9 may be selected with 2 THRU 9, even if elements 3, 5, 6, and 8 do not exist. This is called an open set. Note that large open sets can cause higher computational costs.

**SEUPPLOT****Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

**Format:**

SEUPPLOT seid

**Example:**

SEUPPLOT 7

| Describer | Meaning   |
|-----------|---|
| seid      | Superelement identification number (Integer $\geq 0$ ). |

**Remarks:**

1. See also related command SEPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTS refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

## STEREOSCOPIC

Selects Stereoscopic Projection

See the description of the [ORTHOGRAPHIC, etc., 757](#).

**VANTAGE POINT****Location of the Observer**

Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.

**Format:**

VANTAGE POINT ro so to do sor

**Example:**

VANT 100.

| Describers | Meaning   |
|------------|---|
| ro         | Location of the observer on the r-axis in model units (Real).   |
| so         | Location of the observer and left eye of the observer on the s-axis, in model units, for perspective and stereoscopic projections, respectively (Real).           |
| to         | Location of the observer on the t-axis in model units (Real).   |
| do         | Separation of the observer and the projection plane on the r-axis in model units. The PROJECTION PLANE SEPARATION command may also specify the separation (Real). |
| sor        | Location of the of the observer's right eye for stereoscopic projections in model units (Real).   |

**Remarks:**

1. VANTAGE POINT or the FIND command must be specified if the PERSPECTIVE or STEREOSCOPIC command is also specified.
2. The VANTAGE POINT command is not recommended. The FIND command is recommended because it automatically calculates the optimum vantage point.
3. A theoretical description of the vantage point is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

## VIEW

## Angular Relationship of Observer's Coordinate System

Defines the angular relationship between observer's coordinate system (r, s, and t axes specified on the AXES command) and basic coordinate system.

Format:

VIEW gamma beta alpha

Example 1:

View the model from the r-axis.

VIEW 0. 0. 0.

Example 2:

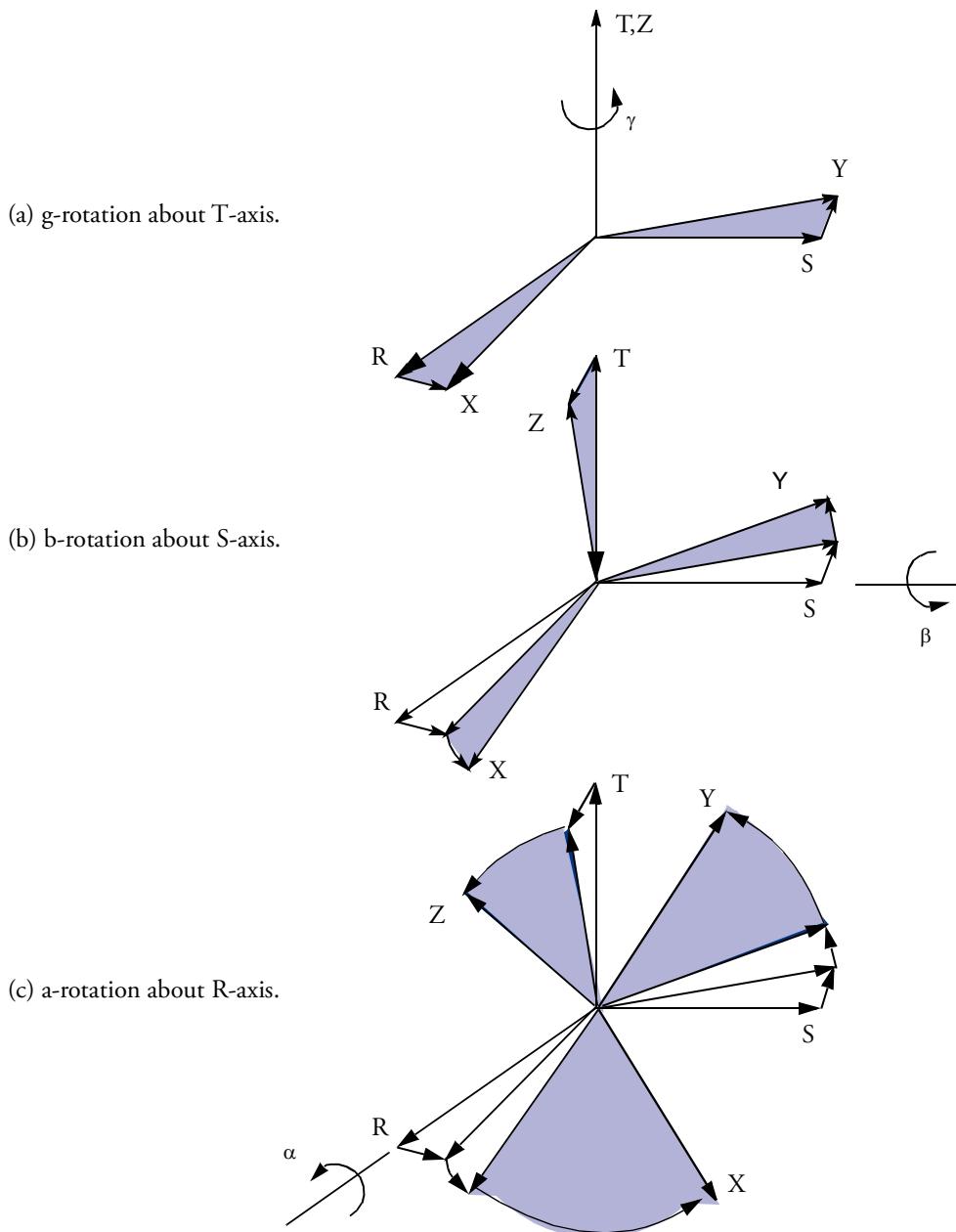
View the model midway between the r- and s-axes.

VIEW 45. 0. 0.

| Describers | Meaning  |
|------------|--|
| gamma      | Angle of rotation, in degrees, of t axis specified on AXES command (Default = 34.27).  |
| beta       | Angle of rotation, in degrees, of s axis specified on AXES command (Default = 23.17 if ORTHOGRAPHIC or STEREOSCOPIC command is specified and 0.0 if PERSPECTIVE command is specified). |
| alpha      | Angle of rotation, in degrees, of r axis specified on AXES command (Default = 0.0).  |

Remarks:

1. If no VIEW command is specified, then VIEW 34.27 23.17 0.0 is assumed for orthographic and stereoscopic projections; and VIEW 34.27 0.0 0.0 is assumed for perspective projections. The default values produce a plot in which unit vectors on the axes of the basic coordinate system have equal lengths.
2. The angles are rotated in sequence: gamma rotates the t-axes, followed by beta which rotates the s-axes, followed by alpha which rotates the r-axes.



3. The VIEW command specifies the position of the model with respect to the s-t plane. Gamma and beta represent the angles of turn and tilt. Alpha is normally not used since it does not affect the orientation of the s-t plane, but only its orientation on the plot frame.



# 6

## Parameters

- Parameter Descriptions 784
- Parameter Applicability Tables 1012

## Parameter Descriptions

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. The PARAM Bulk Data entry is described in the [Bulk Data Entries, 1117](#). The PARAM Case Control command is described in the [PARAM \(Case\), 529](#). PARAMs may also be used in .rc and .ini files as described in [Specifying Parameters, 3350](#). in the *MSC Nastran Installation and Operations Guide*. A complete alphabetical list of PARAMeter names and their functions is given in this section.

If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:

1. PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
2. PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.
3. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.

### **ACEXTMTD**

ACEXTMTD is used to select a different solver for solving exterior acoustic model. Valid options are ITER for selecting iterative solver and KRYLOV for selecting Krylov solver.

### **ACEXTSET**

ACEXTSET is used to select ITER bulk data entry for PARAM,ACEXTMTD,ITER.

### **ACOUT**

Default = PEAK

ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis. ACOUT=RMS requests root-mean-square output.

To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

$$\text{dB} = 20 \cdot \log\left(\frac{P}{\text{PREFDB}}\right)$$

Instead of ACCELERATION, suffix INT can be utilized to generate INTENSITY. See also the Case Control command [FLSTCNT \(Case\), 379](#).

### **ACOWEAK**

Default = NO

ACOWEAK controls the execution of weakly coupled acoustic formulation logic. PARAM,ACOWEAK,YES will activate weakly coupled acoustic formulation which will solve exterior acoustic model along with CACINFx separately using load generated from structural response and coupling matrix. ACOWEAK,YES is available only in SOL 111 and SOL 200 with ANALYSIS=MFREQ.

In the cases where loading is on exterior acoustic only, PARAM,ACOWEAK,YESR will solve for exterior acoustic model first and generates loading at coupling nodes for structural and interior acoustic model. In addition, PARAM, ACOWEAK, YESR can be used to force NASTRAN to solve exterior acoustic model first if loading is on both structural and exterior acoustic models.

Note that the detailed knowledge on the loading is not required to use ACOWEAK, and with PARAM, ACOWEAK, yes, the Automated logic will determine which part of FSI to solve first.

The ACOWEAK solution technique supports trim components via TRIMGRP or ACTRIM between structure and interior cavity only. If there are trim components situated between the structure and exterior cavity, fully coupled solution techniques should be utilized.

With ACOWEAK solution scheme, the exterior acoustic portion of the model is solved in physical coordinates. Therefore, acoustic structural modal participation factor will not be possible since acoustic field eigenvalues and/or eigenvectors are missing or incomplete.

## ACSYM

Default = YES

By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM,ACSYM,NO requests the pre-Version 69 formulation which involves no symmetrization and will require more CPU time. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*. See also the Case Control command [FLSTCNT \(Case\), 379](#).

If the iterative solver is selected (see the Case Control command, [SMETHOD \(Case\), 579](#)) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

## ACTDMP

Default=0

This parameter can be used to specify DMP processors for Actran. By default(ACTDMP=0), it will be set to DMP of Nastran job submittal.

## ACTSMP

Default=0

This parameter can be used to specify SMP processors for Actran. By default(ACTSMP=0), it will be set to SMP of Nastran job submittal.

## ACTMEM

Default=0

This parameter can be used to specify TOTAL amount memory for all processors for Actran via DMP or PARAM,ACTDMP in 4 bytes mega-words. By default (ACTMEM=0), Actran get the memory assigned to master node only. An example, for 'memorymax=64gb mem=max', use PARAM,ACTMEM,16000 which get ACTRAN same amount of TOTAL memory as NASTRAN.

## ADJMETH

Default = 0

This parameter selects the processing method used in a triple matrix product in module DSADJ. The default is usually preferred, but ADJMETH=1 can be used when disk space is critical. ADJMETH=2 only holds the active solution vectors.

## **ADMEXTU**

Default = 0

ADMEXTU is a file unit number used by the SOL 111/112 restart run along with ADMPOST=1. This file unit no is assigned to external super element op2 file. See the Case Control command [ADAMSMNF\\* \(Case\)](#), footnote and Remark 20.

## **ADMPOST**

Default = 0

This parameter allows the user to bring an ADAMS results file into Nastran for modal data recovery in SOL 111 and SOL 112. A value of 1 brings in the ADAMS results without consideration of rigid body motion in the display of the results. A value of 2 brings in the ADAMS results with consideration of rigid body motion in the display of the results. See the Case Control command [ADAMSMNF\\* \(Case\)](#), 221 footnote and Remark 19. for more details on restrictions and required file assignments.

## **ADPCON**

Default = 1.0

Initial penalty values used in contact analysis are calculated automatically by the program and are given by  $k \cdot SFAC \cdot |ADPCON|$  where k is a number selected for each slave node based on the diagonal stiffness matrix coefficients that are in the contact region, and SFAC is the value specified by the user in the SFAC field of the BCONP Bulk Data entry. The ADPCON value applies to all the contact regions in the model. During the analysis, if convergence problems are encountered, penalty values are automatically reduced. Still there may be some problems where convergence can not be achieved with default values. In such cases, analysis may be restarted with a lower value of ADPCON.

In some cases the default penalty values may be low. In such situations analysis may be restarted with a higher value of ADPCON.

Generally, penalty values are recalculated every time there is a change in stiffness. However, if ADPCON is negative, penalty values are calculated only at the beginning of each subcase, and penalty values are not adjusted during analysis. This is useful if the contact between two elastic bodies is being analyzed.

## **ADSTAT**

Character; Default = YES

In transient analysis (SOLs 101,109 and 112 only), if there is a preload (see Case Control command [STATSUB \(Case\)](#), 585) and ADSTAT=YES, the static displacements, SPC forces, and MPC forces will be included in the transient output. There is a limitation that OLOAD is not included. Another limitation is that element stress and force recover will neglect thermal strains and, hence will be wrong if there is a TEMP(LOAD) request. For any other value of ADSTAT the preload effects will not be included in the output, hence the output will be the perturbation displacements, SPC forces and MPC forces.

## AERODOF

Default=35

Digits 1 through 6 indicate which k-set degrees of freedom are to be output. This is only for Double Lattice lifting surfaces that provide output in the plunge (3) and pitch (5) degrees of freedom by default. If additional digits are specified (e.g. AERODOF=1235), then the k-set aerodynamic matrices are padded with null columns for the additional degrees of freedom. Nastran will not create aerodynamic data for these added degrees of freedom. This parameter is only useful to the user that knows how to augment the existing aerodynamic data from an external source. Degrees of freedom 3 and 5 may not be omitted.

## AESDISC

Default = 1.E-8

Tolerance for discarding generalized coordinates in the RITZ method (see Case Control command, [AESMETH, 787](#)) which are not linearly independent.

## AESMAXIT

Default = 15

Maximum number of iterations for the ITER method (see Case Control command, [AESMETH, 787](#)).

## AESMETH

Default = SELECT

Solution method for static aeroelastic analysis.

- |        |   |
|--------|---|
| SELECT | selects the DIRECT method on models with less than 50000 DOF in the solution set; otherwise selects AUTO.             |
| AUTO   | selects the reduced basis method for an approximate solution, which is used as starting vectors for an ITER solution. |
| DIRECT | selects the direct solution.  |
| RITZ   | selects the reduced basis approximate solution.   |
| ITER   | selects the iterative solution.   |

## AESRNDM

Default = 2

Number of random vectors to use as generalized functions in the RITZ method (see Case Control command, [AESMETH, 787](#)).

## AESTOL

Default = 1.E-10

Convergence criteria for the iterative solver.

## AGGRROT

Default = YES

PARAM, AGGRROT, YES (Default) produces both translation and rotational terms in the structure-fluid interface matrix AGG. PARAM, AGGRROT, NO produces only translational terms in the structure-fluid interface matrix AGG. Since fluid elements see no rotational degrees of freedom, AGGRROT YES or NO should have no significant effect on the results of the fluid cavity. Results on the structural side may change especially near unconstrained boundaries where there may be a reversal in the signs of some acceleration terms.

## ALPHA1, ALPHA2

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1 is the scale factor applied to the mass matrix and ALPHA2 to the structural stiffness matrix. In SOL400, ALPHA1 and ALPHA2 are supported for both the linear and nonlinear elements. In SOL 129, Alpha1 scales the mass matrix, and ALPHA2 is not used.

$$[B'] = [B] + \text{ALPHA1} \cdot [M] + \text{ALPHA2} \cdot [K]$$

If  $\xi_i$  is the damping ratio for the i-th mode  $\omega_i$  (radians/unit time), then ALPHA1 ( $\alpha_1$ ) and ALPHA2 ( $\alpha_2$ ) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{bmatrix} \omega_j & -\omega_i \\ -1 & \frac{1}{\omega_i} \end{bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other  $\xi_l$  mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[ \left( \frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left( \frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

|              |  |
|--------------|--|
| <b>Note:</b> | <p>The use of Rayleigh damping with non-zero values of ALPHA1 may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.</p> <p>Similarly, the use of Rayleigh damping with non-zero values of ALPHA2 may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.</p> <p>Since Version 2005r3, the ALPHA1, ALPHA2 parameters apply only to stationary structural components. Prior to this version, the parameters applied to both stationary and rotating structural components. With the introduction of Version 2005r3, new parameters ALPHAR1, ALPHAR2 are defined on the RSPINR and RSPINT bulk data entry for the rotating component, see Remark 8. of the DAMPING bulk data description for a comment on this item.</p> <p>Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.</p> |
|--------------|--|

## ALPHA1FL, ALPHA2FL

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1FL and/or ALPHA2FL are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1FL is the scale factor applied to the mass matrix and ALPHA2FL to the fluid stiffness matrix.

$$[B'] = [B] + \text{ALPHA1FL} \cdot [M] + \text{ALPHA2FL} \cdot [K]$$

If  $\xi_i$  is the damping ratio for the i-th mode  $\omega_i$  (cycles/unit time), then ALPHA1FL ( $\alpha_1$ ) and ALPHA2FL ( $\alpha_2$ ) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{Bmatrix} \omega_j & -\omega_i \\ -1 & 1 \\ \omega_j & \omega_i \end{Bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other  $\xi_l$  mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[ \left( \frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left( \frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

|              |   |
|--------------|---|
| <b>Note:</b> | <p>The use of Rayleigh damping with non-zero values of ALPHA1FL may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.</p> <p>Similarly, the use of Rayleigh damping with non-zero values of ALPHA2FL may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.</p> <p>Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.</p> |
|--------------|---|

## ALTRED

Default = NO

ALTRED=YES requests the alternate stiffness and load reduction technique for superelement analysis in SOLs 101 and 114. This technique is described in [Static Solutions in SubDMAP SEKRRS, 375](#), [Static and Dynamic Load Generation, 378](#), and [Data Recovery Operations in SubDMAP SEDISP](#) in the *MSC Nastran Reference Guide*.

## ALTSHAPE

Default = 0

ALTSHAPE selects the set of displacement shape functions to be used in p-version analysis. PARAM,ALTSHAPE,0 selects the MacNeal set. PARAM,ALTSHAPE,1 selects the Full Product Space set. For ALTSHAPE=1, IN=1 and ISOP=1 must be specified on the PSOLID entry.

## ARBMASP

Default = 2

ARBMASP defines the maximum aspect ratio of CQUAD4 generated for CP/OP options of PBMSECT if ARBMSTYP=timoshen. Since the thickness of a ply is usually the small dimension, ARBMASP affects the size of CQUAD4 lengthwise along a segment.

## ARBMFEM

Default = YES

ARBMFEM controls the generation of '.bdf' file which contains the Finite Element Model of the arbitrary beam cross section. This parameter is functional for PBRSECT and PBMSECT only. To turn off the capability, set value of ARBMFEM to 'NO'.

## ARBMNOW

Default = 0

ARBMNOW controls the overwrite of segment property for PBMSECT, arbitrary beam cross section. With default value of ARBMNOW, segment property overwrite will occur during the process of assigning the property for each segment. To turn off segment property overwrite, use PARAM,ARBMNOW,1.

## ARBMP

Default = YES

ARBMP controls the generation of outline plot for arbitrary beam cross section in PostScript format. This parameter is functional for PBRSECT and PBMSECT only. To turn off capability, set value of ARBMP to 'NO'.

## ARBMS

Default = NO

ARBMS controls the stress recovery for the whole arbitrary beam cross section and the companion 'screened' stresses. The stress recovery for the whole cross section is available in 'OP2' format and suitable for post-processing. The 'screened' stresses for CBAR and CBEAM elements is available in print file (f06) and can be utilized for design optimization via RTYPE=ABSTRESS on DRESP1. This parameter is functional for PBRSECT and PBMSECT only. To turn on the capability, set value of ARBMS to 'YES'.

**Note:** The recovery of 'screened' stresses will be turned on automatically if RTYPE=ABSTRESS is in use on DRESP1.

## ARBMSTYP

Default = TIMOSHEN

ARBMSTYP controls the solution algorithm for composite arbitrary beam cross-section. Other valid option is 'TIMOFORC'. This parameter is functional for PBRSECT and PBMSECT only.

## ARF

Default = 0.95

See [FLUIDMP, 828](#).

## ARS

Default = 0.95

See [FLUIDMP, 828](#).

## ASCOUP

Default = YES

In coupled fluid-structure analysis, by default, the coupling between the fluid and structure is computed. This interaction will be ignored if PARAM,ASCOUP,NO is specified. See also the Case Control command [FLSTCNT \(Case\), 379](#) for alternative selections.

## ASING

Default = 0

ASING specifies the action to take when singularities (null rows and columns) exist in the dynamic matrices (or  $[K_{ll}]$  in statics). If ASING=-1, then a User Fatal Message will result.

If ASING=0 (the Default), singularities are removed by appropriate techniques depending on the type of solution being performed. The procedures used are described in [Data Recovery Operations in SubDMAP SEDRCVR, 392](#) and [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide*.

## AUNITS

Default = 1.0

AUNITS is used in SOL 144 to convert accelerations specified in units of gravity on the TRIM Bulk Data entry to units of distance per time squared. Accelerations are divided by this parameter. To convert accelerations input in g's into physical, consistent units, set AUNITS to 1/g.

## AUTOADJ

Default = YES

In SOL 200, a value of yes will automatically choose the direct or adjoint sensitivity analysis based on the performance criteria. Other options for this parameter are:

NO - the adjoint method will not be selected.

NOSTAT - the adjoint method will not be selected for static analysis subcases, but may be selected for frequency response subcases based on the performance criteria.

NOFREQ - the adjoint method will not be selected for frequency responses subcases, but may be selected for static analysis subcases based on the performance criteria.

The default should be preferred in all cases. However, the other options allow investigation of the alternative of using direct sensitivity methods.

## AUTOGOUT

Default = NO

PARAM,AUTOGOUT,YES simplifies grid point data recovery requests when only a set of elements is specified. In other words, given a set of elements the program will automatically determine all connected grid points and honor the data recovery requests for both grid points and elements. See also PARAM,OELMOPT description.

If PARAM,AUTOGOUT,YES is specified then the program assumes that SET 2 defines the set of desired elements and SET 1 defines the additional grids **not** connected to the elements in SET 2. If no additional grids are desired then the user need only specify SET 1=0. For example, in Case Control,

```
Param,autogout,yes
Set 2 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 1 = 0                                     $ additional grids
Displ=1
Gpfor=1
Stres=2
```

If the user prefers to use different set IDs then user PARAMs OSETELE and OSETGRD may be used to change the set ids used by this feature. For example:

```
Param,autogout,yes
Param,osetele,200
Param,osetgrd,100
Set 200 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 100 = 0                                     $ additional grids
Displ=100
Gpfor=100
Stres=200
```

And if the user wants to add more grids that are not connected to the element set:

```
Param,autogout,yes
Param,osetele,200
Param,osetgrd,100
Set 200 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 100 = 29                                     $ additional grids
Displ=100
Gpfor=100
Stres=200
```

## AUTOMSET

Character, Default = NO (Except in SOL 400 with ANALYSIS=STATICS)

The relationship between dependent and independent degrees-of-freedom for rigid elements and MPCs may be altered from the user's input specification via PARAM,AUTOMSET,YES or PARAM,AUTOMSET,LUSOL. For PARAM, AUTOMSET,YES, or PARAM,AUTOMSET, LUSOL, dependent

degrees-of-freedom of the model, the M-set are automatically determined by employing a rectangular decomposition of the RMG matrix.

|       |  |
|-------|--|
| YES   | For solutions other than SOL 600, the rectangular decomposition is performed via UMFPACK, which is used by permission. Copyright (c) 2002 by Timothy A. Davis. All rights reserved. Availability: <a href="http://faculty.cse.tamu.edu/davis/suitesparse.html">http://faculty.cse.tamu.edu/davis/suitesparse.html</a>  |
| LUSOL | The rectangular decomposition is performed via LUSOL, which is capable of Rook or full pivoting during rectangular decomposition. LUSOL option is especially suitable for ill conditioned RMG matrix or problems with redundant multi-point constraints. Availability: <a href="http://web.stanford.edu/group/SOL/software/lusol/">http://web.stanford.edu/group/SOL/software/lusol/</a> |

**Notes:**

- When used in SOL 600, AUTOMSET=Yes maps to Marc's AUTOMSET. By default or if PARAM,AUTOMSET,NO is entered, Marc's AUTOMSET will be omitted. See Volume C of Marc for further details.
- PARAM,AUTOMSET,YES is not supported for SOL 200.
- RSSCON is not supported with AUTOMSET.

**AUTOQSET**

Default = NO

AUTOQSET=YES requests the automatic calculation of component modes without the need to define a q-set (generalized coordinates).

1. The calculation of component modes is attempted on all superelements including the residual structure.
2. All component modes are treated like SPOINTs which means that all component modes are “brought down” to and assigned to the q-set in the residual structure. In other words, component modes may not be assigned interior to a superelement.
3. Selected component modes may not be removed (constrained).
4. Since the generalized coordinates are automatically defined, the following entries may not be specified: QSETi, SEQSETi, SENQSET, or PARAM,NQSET.
5. This feature is not supported with:
  - a. Design optimization (SOL 200)
  - b. Aerodynamic analyses (SOLs 144, 145, 146)
  - c. Cyclic symmetry analyses (SOLs 114, 115, 116, 118)
  - d. SECSETi and SEBSETi Bulk Data entries that result in a fixed-free or free-free superelement. This restriction also applies to BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1entries in part (BEGIN SUPER) superelements.

PARAM,SQSETID may be used to specify the starting identification number of the automatically generated q-set degrees-of-freedom except when the EXTSEOUT command is present (see Remark 17. of the EXTSEOUT Case Control command). The default is 99000001.

## AUTOSPC

Default = YES

This parameter is obsolete and replaced by the Case Control command [AUTOSPC \(Case\), 249](#). It is ignored for the residual structure in all nonlinear analysis solution sequences. SOL 400 only support the AUTOSPC Case Control command. For the residual structure in SOLs 106 and 129 see the description of parameter AUTOSPCR. AUTOSPC and the related parameters EPPRT, EPZERO, PRGPST, and SPCGEN are analogous to and described under the AUTOSPC Case Control command under the EPSING, EPS, PRINT, and PUNCH keywords, respectively. The specification of the AUTOSPC Case Control command overrides the specification of PARAM,AUTOSPC and its related parameters EPPRT, EPZERO, PRGPST, and SPCGEN.

## AUTOSPCR

Default = NO (SOLs 106 and 129 only)

In SOLs 106 and 129 only, AUTOSPCR specifies the action to take when singularities exist in linear stiffness matrix of the residual structure after multipoint constraints are processed. AUTOSPCR=YES means that singularities will be constrained and AUTOSPCR=NO means they will not be constrained. It is recommended that all degrees-of-freedom attached to nonlinear elements be specified on ASETi entries. Parameters EPPRT, EPZERO, PRGPST, and SPCGEN may be used with AUTOSPCR.

## AUTOSPRT

Default = YES

By default, free-free models will be automatically constrained for calculation of residual vectors (RESVEC Case Control command) as long as  $F1 \leq 0.0$  on the EIGR (or EIGRL) Bulk Data entry. The auto-SUPPORT method may be deactivated by specifying a SUPPORTi entry, PARAM,AUTOSPRT,NO, or  $F1 > 0.0$ .

FZERO is the maximum frequency assumed for a rigid body mode. FZERO is used by the auto-SUPPORT method to extract the rigid body frequencies. The default is 1.0.

## BAILOUT

Default = 0

See [MAXRATIO, 893](#).

## BEAMBEA

Real, Default = 1000.0

Value      Equivalent radius to be used for beam-beam contact problems. For tubes or round bars, use the outer radius. If the radii are different enter the largest outer radius. For beams, enter an equivalent radius calculated as follows:

$$I=0.5*(Ix+Iy)$$

$$R=\sqrt{A/\pi^2+2*I/A}$$

where A, Ix, Iy are the cross-section properties and  $\pi^2 = \pi^2$ .

## BEIGRED

Default = YES

PARAM,BEIGRED,YES requests a more efficient method to reduce viscous damping matrices. CPU and disk space savings may be significant if there is a small number of viscous damping elements. BEIGRED is automatically disabled if the number of DOF connected to viscous dampers exceeds the value specified by PARAM,MAXDAMP.

MAXDAMP (default=1000) specifies the maximum number of DOF connected to viscous damping elements, such that an alternate reduction method is employed. When there are relatively few viscous dampers, an alternate reduction method is chosen automatically for efficiency. The efficiency is lost however if the number of viscous damping DOF exceeds about 1000. If the number of viscous damper DOF exceeds MAXDAMP, User Warning Message 9166 is printed and the alternate damping reduction logic is disabled. Also note that if MAXDAMP is exceeded, FASTFR is disabled.

## BIGER, BIGER1, BIGER2, BIGER3

Default = 0.0

See S1.

## BUCKLE

Default = -1

BUCKLE=1 requests a nonlinear buckling analysis in a restart run of SOLs 106 or 153. See the *MSC Nastran Handbook for Nonlinear Analysis*. (Not supported for SOL 600.)

BUCKLE=2 requests buckling in a SOL 106 cold start run. (Must be in Bulk Data for SOL 600.)

## CASIEMA

Default = NO, SOL 400 CASI Solver only

When the CASI element-based iterative solver is specified (see the Case Control command [SMETHOD \(Case, 579\)](#)) SOL 400 skips various processes associated with the global structure stiffness matrix. The absence of a complete assembled stiffness matrix and its sub-sets can lead to a termination of the solution process in the NLSOLV module if the CASI solver detects a singularity. If the parameter is set to YES, SOL 400 performs all assembly and displacement set reduction operations involving the complete stiffness matrix allowing the

NLSOLV module to select either the matrix-based iterative solver or the sparse direct solver to continue in the event CASI detects a singularity.

Use a value of YES with caution as it adversely affects run-time performance. Substantial additional disk space could be required as well as longer run times. Keep in mind that invocation of the sparse direct solver for large solid element models may degrade performance even further.

This parameter only applies to SOL 400 usage of the CASI solver.

## CASIMEST

Default = Yes

This parameter controls the memory estimation inside CASI iterative solver. (SOL 101, SOL 400 and SOL 200)

= No; Turns off memory estimation.

## CASPIV

Default = 1.0e-10

Input DMAP parameter to the SOLVIT and NLSOLV modules for the CASI iterative solver. It is the pivot threshold for the CASI PCG (Preconditioned Conjugate Gradient) factorization. Pivot value less than CASPIV will exit with an error message.

## CB1, CB2

Default = (1.0, 0.0)

CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is:

$$[B_{jj}] = CB1 \cdot [B_{jj}^x] + CB2 \cdot [B_{jj}^2]$$

where  $[B_{jj}^2]$  is selected via the Case Control command B2GG and  $[B_{jj}^x]$  comes from CDAMPi or CVISC element Bulk Data entries. These parameters are effective only if B2GG is selected in the Case Control Section.

## CDIF

Default = YES for shape optimization with or without property optimization.

Default = NO for property optimization only.

CDIF may be used to override the default finite difference scheme used in the calculation of pseudo loads in SOL 200. PARAM,CDIF,YES forces the selection of the central difference scheme used in the semianalytic approach regardless of the type of optimization requested. PARAM,CDIF,NO forces the selection of the forward difference scheme.

## CDITER

Default = 0

If CDITER > 0, perform constrained displacement iterations in SOL 101. The value is the maximum number of iterations. If CDPRT=YES, print those negative displacements and tension forces which do not satisfy constraints. If CDPCH=YES, punch DMIG CDSHUT entries for final state; by default all gaps are closed. These can be used for initial conditions for restart. Potential contact points must be specified on the SUPORTi entries. The SUPORTi points must be in the residual structure. Optional DMIG entries to define the initial shut vector may be specified. Degrees-of-freedom that are specified on the SUPORT entry and have a value of 1.0 defined on the DMIG,CDSHUT entry will be considered closed initially.

If the DMIG,CDSHUT entry is not supplied, then all degrees-of-freedom specified on the SUPORT entries will be considered shut initially. A fatal message will be issued if this parameter is used and PARAM,INREL is specified.

### **CDPCH**

Default = NO

See [CDITER, 797](#).

### **CDPRT**

Default = YES

See [CDITER, 797](#).

### **CFDIAGP**

Default = NO

If YES, randomly deleted CFAST elements will be printed. (See CFRANDEL)

### **CFRANDEL**

Default = 0.

Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted.

### **CHECKOUT**

Default = NO

CHECKOUT=YES requests a model checkout in SOLs 101 through 200. See [Geometry Processing in SubDMAP PHASE0](#) in the MSC Nastran Reference Guide. The run will terminate prior to phase 1 of superelement analysis. The PARAM,POST options are also available with PARAM,CHECKOUT,YES. The following options and their user parameters are also available with PARAM,CHECKOUT,YES:

1. PARAM,PRTGPL,YES

Prints a list of external grid and scalar point numbers in internal sort. It also lists external grid and scalar point numbers along with the corresponding sequence numbers in internal sort. The sequence numbers are defined as (1000\*external number) and will reflect any user-requested resequencing.

2. PARAM,PRTEQXIN,YES

Prints a list of external and internal grid and scalar numbers in external sort. It also lists external grid and scalar numbers with the corresponding coded SIL number in external sort. The coded SIL numbers are defined as:

$$10 \cdot \text{SIL} + \begin{cases} 1 & \text{for grid point} \\ 2 & \text{for scalar point} \end{cases} \quad (1)$$

The SIL numbers correspond to degrees-of-freedom, i.e., one SIL number for scalar point and six SIL numbers for a grid point.

### 3. PARAM,PRTGPDT,YES

Prints, for each grid and scalar point, the following information in internal sort:

- Coordinate system ID in which grid point geometry is defined (ID=-1 for scalar points).
- Spatial location of grid points in the “CP” coordinate system. For scalar points, all entries are zero.
- Coordinate system ID for grid point displacements, forces, and constraints (ID=0 for scalar points).
- Permanent single-point constraints defined on GRID Bulk Data entries. A zero is entered for scalar points.

### 4. PARAM,PRTCSTM,YES

Prints for each coordinate system type the transformation matrix from the global to the basic coordinate system, and the origin of the indicated coordinate system in the basic coordinate system. Coordinate system types are: 1 = rectangular; 2 = cylindrical; 3 = spherical.

### 5. PARAM,PRTBGPDT,YES

Prints all grid and scalar points listed in internal sort with their x, y, and z coordinates in the basic coordinate system. In addition, the coordinate system ID for grid point displacements, forces, and constraints is indicated for each grid point (ID=-1 for scalar points). The x, y, and z coordinates of scalar points are zero.

### 6. PARAM,PRTGPTT,YES

Prints, for each temperature load set, information on element and grid point temperatures.

### 7. PARAM,PRTMGG,YES

Prints the g-size mass matrix labeled by grid point/degree-of-freedom.

### 8. PARAM,PRTPG,YES

Prints the g-size load vectors labeled by grid point/degree-of-freedom.

### 9. The summation of forces and moments of applied loads in the basic coordinate system is automatically output for each loading condition requested in the Case Control Section. Related to parameters GPECT and PROUT, and Case Control command ELSUM.

## CK1, CK2

Default = (1.0, 0.0)

CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K_{jj}^x] = CK1 \cdot [K_{jj}^z] + CK2 \cdot [K_{jj}^2]$$

where  $[K_{jj}^2]$  is selected via the Case Control command K2GG and  $[K_{jj}^z]$  is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in Case Control. A related parameter is CK3.

**Note:** Stresses and element forces are not factored by CK1, and must be adjusted manually.

## CK3

Default = (1.0, 0.0)

CK3 specifies a factor for the stiffness derived from GENEL Bulk Data entries. The total stiffness matrix is

$$[K_{jj}] = [K_{jj}^x] + CK3 \cdot [K_{jj}^y]$$

where  $[K_{jj}^y]$  comes from the GENEL Bulk Data entries and  $[K_{jj}^x]$  is derived using PARAMs CK1 and CK2. CK3 is effective only if GENEL entries are defined. Related parameters include CK1 and CK2.

## CLOSE

Default = (1.0)

A value of 1.05 will set natural frequencies within 5% of each other as close. See SCRSPEC.

## CM1, CM2

Default = (1.0, 0.0)

CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] = CM1 \cdot [M_{jj}^x] + CM2 \cdot [M_{jj}^2]$$

where  $[M_{jj}^2]$  is selected via the Case Control command M2GG and  $[M_{jj}^x]$  is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

## COMPMMAT

Default = NO, SOL 106 and SOL 400 only

In nonlinear statics (SOLs 106, 400 ANALYSIS=NLSSTAT), composite materials compute temperature-dependent properties for the plies only at the reference temperature given on the PCOMP Bulk Data entry. The ply properties are smeared and used for all load steps, regardless of whether the temperature is changing through application of thermal loads.

If the parameter is set to YES, the temperature-dependent properties for the plies are updated and smeared at the current temperature for each load step. For CQUAD4 and CTRIA3 elements with offsets, the large rotation offset method specified by "MDLPRM, OFFDEF, LROFF" in SOL 400 must be used.

If the parameter is set to NONSMEAR, the temperature-dependent properties for the plies are updated at the current temperature for each load step. This option, only available for the CQUADR and CTRIAR elements, is an alternative to the smeared approach.

This parameter only applies to SOLs 106 and 400 ANALYSIS=NSTAT, and only applies to composite CQUAD4, CTRIA3, CQUADR and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

Prior to Nastran version 2019, when COMPMATT was set to YES, material nonlinear was assumed even when no composite plies had temperature-dependent materials or when there was no MATTi type entries associated with temperature dependent plies. This has caused confusion as to when SOL106 or SOL400 is doing linear or nonlinear material solutions. Starting in V2019, PARAM, COMPMATT, YES will only turn on material nonlinear if there are active MATTi entries as well.

Additionally, prior to Nastran version 2019, when PARAM, COMPMATT, YES was used in conjunction with TEMPP1 entries, ply stress computation used only the reference plane temperature. Starting in V2019, ply stress computation will use reference plane temperature plus the thermal gradient term applied at the center of each ply. (To revert to the old method, NASTRAN SDRCMPTP=1 may be added to the Nastran section.) The enhanced nonlinear materials (PSHLN1) do not support the TEMPP1 entry.

## CONFAC

Default = 1.E-5

In superelement analysis, CONFAC specifies the tolerance factor used in checking the congruence of the location and displacement coordinate systems of the boundary points between image superelements and their primaries (see the Bulk Data entry, [CSUPER, 1659](#)). Specification of this parameter is recommended instead of DIAG 37 (DIAG 37 ignores User Fatal Messages 4277 and 4278).

## COSUBCYC

Default = 1, SOL 700 only

Controls the growth of the subcycling interval in the coupling surface.

### Example:

PARAM,COSUBCYC,2

### Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, COSUBCYC is set to 1, and the current number of time steps between updates of the coupling geometry is 4. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

## COSUBMAX

Default = 0, SOL 700 only

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated.

### **Example:**

PARAM,COSUBMAX,10

### **Remarks:**

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values of PARAM,COSUBMAX should be used.

## **COUPMASS**

Default = -1

COUPMASS > 0      Requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability, as listed in [Table 3-1](#) in the *MSC Nastran Reference Guide*. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CONROD, CQUAD4, CHEXA, CPENTA, CPYRAM, CQUAD8, CROD, CTETRA, CTRIA3, CTRIA6, CTRIAX6, CTUBE.

COUPMASS < 0      (Default) causes the generation of lumped mass matrices (which may include torsion inertia for beam elements, and some coupling if there are beam offsets) for all of the above elements. However, if SYSTEM(414) is greater than zero, (Default = 0) then the lumped mass matrices will contain translational components only for the CBAR and CBEAM elements.

P-elements are always generated with coupled mass and are not affected by COUPMASS.

## **CP1, CP2**

Default = (1.0, 0.0)

The load vectors are generated from the equation

$$\{P_j\} = CP1 \cdot \{P_j^x\} + CP2 \cdot \{P_j^2\}$$

where  $\{P_j^2\}$  is selected via the Case Control command P2G, and  $\{P_j^x\}$  comes from Bulk Data static load entries. These parameters are effective only if P2G is selected in the case control section.

## **CQC**

Default = -1

See [SCRSPEC, 987](#).

## CURV

Default = -1

PARAM,CURV,1 requests that the CTRIA3 and CQUAD4 element stress and/or strain output be computed in a material coordinate system (normal output is in the element or basic coordinate system) and/or to interpolate it to grid points. (CQUAD4 element corner stress output is not supported.)

The integer parameter OG controls the calculation of stress and/or strain data at grid points. If OG is set to -1, the calculation for stresses and/or strain data at grid points is not performed. The default value of zero provides the calculation of these quantities at those grid points to which the selected elements connect.

User parameters S1G, S1M, S1AG, and S1AM, set to 1, request the printout of stresses at grid points, stresses in the material coordinate system, strains at grid points and strains in the material coordinate system, respectively.

The integer parameter OUTOPT may be set in accordance with the below options to select print, punch, and/or plotter output for stress and/or strain data that are computed in user-defined material coordinate systems.

| OUTOPT Value | Description   |
|--------------|---|
| 0            | Default-standard MSC Nastran device codes are used. |
| 1            | Print only  |
| 2            | Plot only   |
| 4            | Punch only  |

The above values may be combined additively to select two or more forms of output. For example, OUTOPT=6 requests both plot and punch output. Related parameters include BIGER, CURVPLOT, DOPT, NUMOUT, NINTPTS, S1G, S1M.

For stress and/or strain/curvature output in a user-defined material coordinate system MCSID must be defined on MAT1 and MAT2 Bulk Data entries. The values of MCSID reference CORDiR, CORDiC, and CORDiS Bulk Data entries. A value of zero for MCSID does not imply the basic coordinate and will eliminate all elements which reference the MATi from the subject calculations.

1. If these data are requested at the element centers, the program will compute the unit vector  $i_m$  along the T1 or x-axis of the material coordinate system, and compare

$$|\bar{n} \cdot i_m|$$

for each element that references the material coordinate system, where n is the normal to the surface of the element. If

$$|\bar{n} \cdot i_m|^2 \geq .4$$

the projection of the y-axis on the surface of element is taken as the reference axis. Otherwise, the projection of the x-axis on the surface of the element is taken as the reference axis. The angle between the x-axis of the element coordinate system and the projection of the selected reference axis of the material coordinate system is used to transform the stress and/or strain data into the material coordinate system at the element centers.

2. If, on the other hand, the user requests these data at the grid points to which the elements connect the program will interpolate the results from (a) to the grid points to which the elements connect. The parameter NINTPTS=N, the stress and/or strain data at the N closest element centers to the grid point in question will be used in the interpolation. The program may include more than N points in the interpolation if the distance of other element centers is not more than 10% greater than the closest N element centers.

The following specifies the output headings for stresses and/or strains in the material coordinate system.

### **Element stresses (PARAM,S1M,1)**

1. Available in CQUAD4 and CTRIA3 elements
2. Page headings:  
 STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)  
 STRESSES IN TRIANGULAR ELEMENTS (CTRIA3)
3. Under the column FIBER DISTANCE:  
 Z1 is replaced by MCSID.  
 Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

### **Grid point stresses (PARAM,S1G,1 and PARAM,OG,1)**

1. Available for CQUAD4 and CTRIA3 elements
2. Page heading:  
 STRESSES AT GRID POINTS
3. Under the column are:  

$$\left\{ \begin{array}{c} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{array} \right\}$$
 Z1 is replaced by MCSID.  
 Z2=A+10\*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

### **Element strains (PARAM,S1AM,1)**

1. Available for CQUAD4 and CTRIA3 elements
2. Page headings:

STRAINS IN QUADRILATERAL ELEMENTS (CQUAD4)

STRAINS IN TRIANGULAR ELEMENTS (CTRIA3)

3. Under the column FIBER DISTANCE:

Z1 is replaced by MCSID.

Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

### **Grid point strains (PARAM,S1AG,1 and PARAM,OG,1)**

1. Available for CQUAD4 and CTRIA3 elements.

2. Page heading:

$Z2=A+10^*N$  where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

STRAINS AND CURVATURES AT GRID POINTS

3. Under the column are:

$$\left\{ \begin{array}{c} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{array} \right\}$$

Z1 is replaced by MCSID.

### **CURVPLOT**

Default = -1

PARAM,CURVPLOT,1 requests that x-y (or curve) plots whose abscissas are a sequence of grid points and whose ordinates may be displacements, loads, SPC forces, or grid point stresses. To obtain stress plots, set the CURV parameter to +1. The default for DOPT is the length between grid points.

Specify the XYOUTPUT Case Control command in the usual manner, replacing the point ID with the SID of SET1 Bulk Data entries.

The SET1 Bulk Data entries must contain unique SIDs for each set of grid points to be plotted.

User requests for xy-plots of output quantities appear in the Case Control Section in the standard form. For example,

```
.
.
.
OUTPUT (XYOUT)
.
.
XYPLOT DISP 1/4 (T3)
.
.
XYPLOT SPCF 2/5 (T1)
.
.
BEGIN BULK
```

The first XYPILOT command will produce an xy-plot from the displacement output of subcase 1. The abscissa of the curve will reflect the grid point IDs listed on the SET1 entry with an SID of 4, and the ordinate will reflect the T3 component of displacement at these grid points. The second XYPILOT command will produce an xy-plot whose ordinates are the T1 components of the forces of constraint in subcase 2 at the grid points listed on the SET1 entry with an SID of 5.

The user has some degree of control over the scaling of the abscissas on these xy-plots. This control is exercised through the parameter DOPT on a PARAM Bulk Data entry. The legal values of this parameter provide the following scaling options for the abscissas.

| Value of DOPT  | Scaling for Abscissa |
|----------------|----------------------|
| 0<br>(Default) | $\ g_j - g_i\ $      |
| 1              | $ x_j - x_i $        |
| 2              | $ y_j - y_i $        |
| 3              | $ z_j - z_i $        |
| 4              | 1.                   |

Thus, the default value of DOPT will place the first grid point listed on the referenced SET1 entry at the origin, and subsequent grid points will be located along the abscissa at intervals proportional to the distance between that grid point and its predecessor. Values of DOPT equal to 1, 2, or 3 will scale the abscissa so that the interval between adjacent grid points is proportional to the difference in the X, the Y, and the Z components of the subject grid points respectively. DOPT=4 will space the grid points equally along the abscissa.

## CWDIAGP

Default = NO

For CWELD element: prints elements randomly deleted if set to yes.

## CWRANDEL

Default = 0.0

For CWELD element: if not zero, then it specifies as a decimal percent for the number of fasteners to randomly delete.

## DBALL

Default = DBALL

By default, all data to be stored on the database for restart purposes will be located on the DBALL database set DBset). These parameters permit the storage of some data blocks on DBsets other than DBALL, which are defined by the user and specified on the INIT File Management statement. Any or all of these parameters may be set to SCRATCH in order to reduce overall disk space usage; e.g., PARAM,DBUP,SCRATCH or PARAM,DBALL,SCRATCH. However, automatic restarts will be less efficient because data normally assigned to a permanent DBset will have to be recomputed.

A unique value for each superelement may be specified in the Case Control Section for the parameters DBALL, DBDN, DBRCV, and DBUP. Certain DBsets may be taken offline depending on which phase (see [Summary of Solution Sequence Operations](#) in the *MSC Nastran Reference Guide*) of superelement analysis is being performed. PARAM,DBALL specifies the default value for parameters DBDN, DBUP, and DBRCV.

The DBDN DBset contains data blocks necessary for “downstream” processing. For example, the stiffness, mass, damping, and static loads matrices that have been reduced to the boundary of the superelement are stored in this DBset.

The DBRCV DBset contains data blocks that must be online during the first pass through data recovery (Phase 3). These data blocks are used to recover the total displacement vector  $u_g$  of the superelement. This operation is performed by the SSG3 and SDR1 modules. On subsequent data recovery restarts, this DBset may be taken offline. Its default is determined from the value of DBUP.

The DBUP DBset contains data blocks necessary for “upstream” processing. For example, the geometry and property tables along with the stiffness, mass, damping, and static loads matrices related to the interior grid points of the superelement are stored in this DBset. These matrices and tables must be online during the reduction (Phase 1) and data recovery (Phase 3) of the superelement.

The IFP DBset contains data blocks that are required for all phases of the analysis. These data blocks are related to the entire model; examples are Bulk Data, superelement map, IFP module outputs, and resequenced grid points. This DBset must be online for all runs.

PARAM,DBALL also specifies the default value for PARAMs DBEXT and DSO described below.

### **DBCCONV**

Default = XL

See POST=0.

### **DBCdiag**

Default = 0

See POST=0.

### **DBCOVWRT**

Default = YES

See POST.

### **DBDICT**

Default = -1

Controls the printout of the database directory at the beginning and end of the run. See DBDICT FMS statement description in Section 2. If DBDICT=0, then the database directory will be printed at the start of the run. If DBDICT=1, then the directory will be printed at the end of the run. If DBDICT $\geq$ 2, then it will be printed at the beginning and end of the run. If DBDICT=-1 (the default), the directory is not printed at the beginning or end of the run.

If multiple versions and/or projects exist on the database, then the parameters DBDRPRJ and DBDRVER allow the user to select the desired project and version, respectively. The appropriate values may be found in

the Project/Version Table that is printed upon restart. If DBDRVER=0 (or DBDRPRJ=0), then the current version (or project) is selected. If DBDRPRJ=-1 (or DBDRVER=-1), then all projects (or versions) are selected.

## **DBDN**

Default = value of PARAM,DBALL.

See DBALL.

## **DBDRPRJ**

Default = 0

Specifies the desired project-identification number. See DBDICT.

## **DBDRVER**

Default = 0

Specifies the desired version-identification number. See DBDICT.

## **DBEXT**

Default = value of PARAM,DBALL

Specifies the DBset location to store the external superelement information. External superelement information is generated by the EXTSEOUT Case Control command and the user PARAMeter EXTOUT.

## **DBRCV**

Default = value of PARAM,DBUP.

See DBALL.

## **DBUP**

Default = value of PARAM,DBALL.

See DBALL.

## **DDRMM**

Default = 0

DDRMM is only recognized if PARAM,SPARSEDRL,NO is specified.

By default, the matrix method of data recovery is used in the modal transient and frequency response solutions. DDRMM=-1 will force calculation of complete g-set solution vectors by the mode displacement method and is required for FATIGUE analysis using SOL 108 or SOL 111 if PARAM,SPARSEDRL,no is also specified.

## **DELCLUMP**

Default = 0.5, SOL 700 only

**Format:**

PARAM,DELCLUMP,VALUE

**Example:**

PARAM,DELCLUMP,0.1

Prevents small clumps in the Euler mesh from determining the time step and prevents the leakage of small masses to isolated regions.

VALUE      Value of DELCLUMP. See Remark 1. (Real  $\geq 0.0$ )

**Remarks:**

1. Material in Eulerian elements of a clump with:  
 $f_{\text{func}} < \text{DELCLUMP} \cdot f_{\text{blend}}$   
is eliminated
2. See also parameter FBLEND.

**DESPCH**

Default = 0

For sizing and shape topography optimization, DESPCH specifies in SOL 200 when the optimized (updated) bulk data entries are written to the PUNCH file. Currently, all the property entries, material entries, and connectivity entries that can be designed and DESVAR, DRESP1, and GRID entries can be written. Notice that the DRESP1 entries will be written if only when a mode tracking is performed and the DRESP1 responses have type FREQ or EIGN.

For topology/topometry optimization, DESPCH specifies when the topology optimized element density values (or topometry optimized values) are written to the element result file jobname.des. This file can be directly read in PATRAN or third party post-processor to displace and animate the topology/topometry optimization results.

|     |  |
|-----|--|
| < 0 | Never  |
| = 0 | at the last design cycle only (Default)  |
| > 0 | at every design cycle that is a multiple of DESPCH and the last design cycle. For example, if n=2 and the maximum number of design cycles is 5 (DESMAX=5 on the DOPTPRM entry), then, DESVAR and GRID entries at design cycle 2, 4, and 5 are written in the punch file. |

**DESPCH1**

Default = 6

DESPCH1 specifies in SOL 200 the amount of data to be written to the .pch and .des file. A positive DESPCH1 value request large field formats while a negative value requests small field formats. For a shape

optimization job, if DESPCH1<>0, the updated GRID entries of the whole model will be written in the .pch file.

Descriptions of various DESPCH1 values are given below:

- 0 Write no data.
- $\pm 1$  Write the property entries that are designed.
- $\pm 2$  Write all the property entries of a given type when one or more property of that type is designed.
- $\pm 4$  Write DESVAR and DRESP1 entries.
- $\pm n$  Write combine quantities by summing the DESPCH1 values. For example,  $n=1+4=5$  requests writing all the designed property entries, DESVAR and DRESP1 entries to the .pch file for normal modes analysis.
- $> 0$  Write all (topology designed and non-designed) element density values to the topology element density history file jobname, des.
- $< 0$  Write topological designed element density values to the topology element density history file jobname.des.

## DFREQ

Default =  $10^{-5}$

DFREQ specifies the threshold for the elimination of duplicate frequencies on all FREQi Bulk Data entries. Two frequencies,  $f_1$  and  $f_2$ , are considered duplicated if

$$|f_1 - f_2| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

## DIROUT

Default = NO

See CQC under [SCRSPEC, 987](#).

## DOPT

Default = 0

See CURVPLOT.

## DPEPS

Default = 1.0E-4

In SOL 200, if any difference between the property value on the property entries and the value calculated from the design variable values on the DESVAR entry (through DVCRELi, DVMRELi, DVPRELi relations) is greater than DPEPS, the design model values override the analysis values. If all the differences are less than DPEPS, analysis results from a previous run are accepted in a subsequent sensitivity/optimization task,

thereby avoiding a reanalysis. The PTOL parameter on the DOPTPRM entry is a related parameter that checks the maximum difference.

## DPHFLG

Integer  $\geq 0$ ; Default = 0

Select the method for performing eigenvector sensitivity analysis. Unless repeated eigenvalues are anticipated, the default value is recommended.

|         |     |   |
|---------|-----|---|
| DPHFLG  | 0   | Nelson's method (Default)   |
|         | 1   | Subspace iteration method. Recommended when there are repeated roots.   |
|         | 2   | High's method, # of modes for iteration = $\min(2n, n+8, m)$<br>n: the highest constrained mode<br>m: the number of modes request by EIGR |
|         | 3   | High's method will all modes requested by EIGR used in the iterator.<br>DPHFLG = 2 and 3 have further control parameters                  |
| ITERATE | yes | Do iteration, for improved sensitivity value. (Default)   |
|         | no  | No iteration, equivalent to Fox's method and provides the most efficient method, but not the most accurate.                               |
| ITMAX   |     | Maximum number of iteration. (Default = 10)   |
| TOL     |     | Tolerance for convergence in iteration. (Default = 1.0e-4)  |
| LAMBDA  |     | Shift factor. (Default = 0.0)   |
| KORTHO  | no  | Use mass for Gram Schmidt orthogonalization. (Default)  |
|         | yes | Use stiffness K for Gram Schmidt orthogonalization.   |
| ITRPRNT | no  | Do not print sensitivity for each iteration. (Default)  |
|         | yes | Print sensitivity for each iteration.   |
| ITFPRNT | no  | Do not print final sensitivity, leave print to SOL 200.   |
|         | yes | Print final sensitivity inside High's method computation  |
| MDOF    | no  | Do not reduce DPHI to USET 'U6' DOF. (Default)  |
|         | yes | Reduce DPHI to USET 'U6' DOF.   |

## DSNOKD

Default = 0.0

DSNOKD specifies a scale factor to the differential stiffness matrix in buckling design sensitivity analysis. If DSNOKD  $> 0.0$ , the effect of the differential stiffness matrix is included in buckling the design sensitivity analysis.

If PARAM,DSNOKD,1.0 is specified in SOL 200, the differential stiffness sensitivity calculation is performed more accurately; i.e., the change in the stiffness matrix due to the changes in the displacements are computed. However, the calculation is more expensive than with PARAM,DSNOKD,0.0.

Non-zero values of PARAM,DSNOKD cannot be used in SOL 200 with multiple buckling design subcases less each subcase contains the same STATSUB command.

## DSO

Default = value of PARAM,DBALL

Specifies the DBset location to store datablocks created for design sensitivity and optimization in SOL 200.

## DSZERO

Default = 0.0

DSZERO specifies the minimum absolute value for the printout of design sensitivities.

## DV3PASS

Integer, Default = 1 for SOL 200 with nonlinear property sensitivity evaluation.

DV3PASS controls the number of optimizer internal cycles must pass before the re-evaluation of sensitivity of nonlinear properties, such as I1/I2/J of PBARL/PBEAML/PBRSECT/PBMSECT. With default value of 1, the sensitivity of nonlinear properties is computed for every internal cycle of optimizer. Setting DV3PASS to 10 causes the sensitivity to be evaluated at optimizer internal cycles of 1, 11, 21 and etc. The sensitivity of nonlinear properties is considered as invariants if a sufficiently large number, such as 100, is assigned to DV3PASS.

## DYNSPCF

Default = NEW

PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPCForce calculations for the linear dynamic solutions: SOLs 103, 107 through 112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPCForce results obtained in versions prior to Version 68.

**Note:** Some versions of Dytran have trouble with permanent constraints and issue a bogus error message in the d3hsp file. This is the case for MSC Nastran 2005 r2.

## EIGFILT

Default=1.0E-13

Threshold for zero-frequency eigenvalue for ACMS VERSION=NEW (module ACMS1).

## ENFMETH

Default = TOTAL or ABS

This parameter controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static-based solution. The default value of TOTAL/ABS implies that the program solves directly, *in one step*, for the TOTAL solution of the dynamic analysis which includes both the static-based solution and the dynamic solution that is relative to the static-based solution. If the value is specified as REL, then the program obtains the total solution of the dynamic analysis *in two steps*, by first solving for the static-based solution and then solving separately for the dynamic solution RELATIVE to the static-based solution. In general, the TOTAL/ABS and REL solution methods both yield essentially the same results.

The TOTAL/ABS solution method is computationally more efficient. This is also the only method that is meaningful and that should be employed when a problem involves the use of NOLINI or NLRGAP entries. An important point to note regarding this method is that, for modal dynamic analysis, residual vectors are absolutely critical in order for this method to get correct answers.

The REL solution method, though less efficient, may be more accurate for transient solutions and for modal frequency response solutions at very low forcing frequencies. Also, for modal dynamic analysis, this method is not as critically dependent on residual vectors as the TOTAL/ABS solution method. In addition, the current implementation of the REL solution method does not support fluid DOFs with enforced motion. So, if fluid DOF are among the DOFs with enforced motion, the job will be switched to the ABS solution method automatically.

It should be emphasized here that PARAM,ENFMETH and PARAM,ENFMOTN are completely separate, distinct and independent of each other and should not to be confused with each other. The former controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used while the latter controls how the results of such an analysis are *output*.

## ENFMOTN

Default = TOTAL or ABS

This parameter controls how the results of the analysis are *output* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static solution. The default value of TOTAL/ABS implies that the output results of the analysis represent the TOTAL solution of the dynamic analysis which includes both the static solution and the dynamic solution that is relative to this static solution. If the value is specified as REL, then the output results represent the dynamic solution RELATIVE to the static solution.

It should be noted that, in general, the static solution mentioned above may *not* be a stress-free solution. Therefore, in general, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL in an SPC/SPCD enforced motion analysis may give *different results not only for displacements, but also for stresses*.

In general, it is not easy to identify all cases wherein the static solution represents a stress-free solution. However, one can give examples of such cases.

Thus, for instance, if the SPC/SPCD enforced motion is specified at a single grid point and there are no constraints at any other grid point, then the static solution will be a stress-free solution.

Similarly, if the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the enforced motion is the same for all of the enforced motion points and there are no other constraints in the model, then all of the enforced motion points will move in unison and the static solution will be a stress-free solution.

In contrast to the above, consider the situation in which the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the specified enforced motion is *different* for the various enforced motion points. In this case, the static solution will *not* be a stress-free solution.

If the static solution represents a stress-free solution, then it can be regarded as the base motion of the model. In such cases, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different* displacements, but the same stresses. Further, in modal dynamic analysis for such cases, the results using PARAM,ENFMOTN,TOTAL (or ABS) will give the *same* results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes included and the results using PARAM,ENFMOTN,REL will give the same results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes excluded.

If the static solution does *not* represent a stress-free solution, , the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different results not only for displacements, but also for stresses*. In this case, the results from the usage of PARAM,ENFMOTN,REL do not have the same significance as in the case of the stress-free static solution except for the fact that these results represent the dynamic solution RELATIVE to the static solution.

**Note:** Currently, the usage of PARAM,ENFMOTN,REL is not supported when PARAM,ENFMETH,ABS is specified (or implied). If this usage is employed, the program terminates the execution with an appropriate fatal message.

## EPPRT

Default = 1.E-8

EPPRT is analogous to and described under the EPSSING keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPPRT.

## EPSILONT

Default = SECANT

In nonlinear statics (SOL 106 and SOL 400), thermal loads are computed using the secant method:

$$\epsilon_T = \alpha_{\text{load}}(T_{\text{load}} - T_{\text{ref}}) - \alpha_{\text{init}}(T_{\text{init}} - T_{\text{ref}})$$

If the parameter is set to INTEGRAL, thermal loads are computed using the integral method:

$$\epsilon_T = \int_{T_{\text{init}}}^{T_{\text{load}}} \alpha(T) dT$$

This parameter only applies to SOLs 106 and 400 ANALYSIS=NLSSTAT, and only applies to CQUAD4, CTRIA3, CQUADR, and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

## EPZERO

Default = 1.E-8

EPZERO is analogous to and described under the EPS keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPZERO.

## ERPC

Default = 1.0

This parameter represents the phase speed of the fluid for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERPREFDB

Default = 1.0

This parameter represents a peak reference ERP value used in the computation of ERP in units of dB. The dB level is defined as:

$$\text{ERPdB} = 10 \log \left( \text{RHOCP} \cdot \frac{\text{ERP}}{\text{ERPREFDB}} \right)$$

This parameter may alternatively be set using the ERP Case Control command.

## ERPRHO

Default = 1.0

This parameter represents the fluid density for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERPRLF

Default = 1.0

This parameter represents a Radiation Loss Factor scale factor for the Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERROR

Default = -1

For SOLs 111-112, when the constraint modes have non-zero generalized force the parameter ERROR set to 0 causes the fatal error exit to be branched over and the dynamic response is computed and output. This option is intended for model checkout work, where bad results are better for diagnostic work than no results at all. This parameter is a traditional feature also used in other, similar circumstances.

## ESLFSAV

Character, Default = NO

ESLFSAV = YES requests that all the intermediate files from an ESLNRO job be saved on disk. The destination of these files can be directed with the 'sdir=' option on a Nastran submittal command line.

## ESLMOVE

Integer, Default = 0

ESLMOVE = 0 selects a move limit scheme that poses restrict lower and upper bounds on design variables during the linear response optimization. The range of the bounds is determined by

$$X_k^L = \max(X_o^L, X_{k-1} - MOVE)$$

$$X_k^U = \min(X_o^U, X_{k-1} + MOVE)$$

$$MOVE = \max(DXMIN, abs(X_{k-1}) \cdot DELXESL)$$

ESLMOVE = 1 selects a move limit scheme that scales back the design move proposed from a linear response optimization. The amount of scaling back is determined by

$$X_k^* = X_{k-1} + (X_{k-1}^I - X_{k-1}) \cdot DELXESL$$

where  $X_k^*$  is the scaled design variable for the k-th design cycle,  $X_{k-1}$  is the design variable at (k-1)th design cycle,  $X_{k-1}^I$  is the proposed design from the linear optimization solution at (k-1)th design cycle.

Parameters DXMIN and DELXESL can be specified on the DOPTPRM entry.

## ESLLCOMP

Default = No

ESLLCOMP selects types of compliance response to be included in the design task. The nonlinear compliance response is defined using a DRESP1 entry with RTYPE=COMP for the ESLNRO topology optimization tasks. As the default, it is computed by the product of the applied nonlinear loads and corresponding nonlinear displacement components. Alternatively, ESLLCOMP=YES selects a linear compliance response that computed as the total work done by the equivalent static loads on the linear system.

## ESLMPC1

Default = 0

This parameter applies only to the ESLNRO jobs with 3D contact. Its default has different meanings depending on the type of contact applications. As the default, for a glued contact ESLNRO job, a linear response optimization task will include a set of MPC entries that are created from the nonlinear analysis. For a touching contact ESLNRO job, the linear response optimization task will not include the MPC entries by default. Setting ESLMPC1 to a positive number will turn on the MPC inclusion.

- 1      uses the MPC entries created from the nonlinear analysis at the converged nonlinear analysis.
- 2      uses the MPC entries created at the beginning of the very first nonlinear analysis.

## ESLOPTEX

Default = 0

This parameter allows the user to perform an ESLNRO job at a targeted exit point. The allowable values of ESLOPTX are listed below with their description.

- 0      Do not exit. Proceed with ESLNRO nonlinear response optimization.
- 1      Exit after the initialization of the analysis and design model but before nonlinear FE analysis begins.
- 2      Exit after nonlinear FE analysis ends.
- 3      Exit after design constraint evaluation and screening.

## ESLPRT

Default = 0

ESLPRT specifies how often the ESLNRO results are printed in the .f06 file and saved in the .xdb file. By default, the program will print the results to the .f06 file at the first and the last design cycles and save the results to .xdb (or .op2) at the first and last design cycles on the disk. (See ESLPRT1 for selection of result contents.)

- > 0      then the results are printed at the first design cycle; at every design cycle that is a multiplier of ESLPRT; and the last design cycle.
- < 0      the no results are printed and saved.

## ESLPRT1

Default = 7

ESLPRT1 specifies what type of results to be written to the .f06 and to .xdb (or .op2). It may take any of the following base values or a combination of these base values:

- 0      write no data
- 1      write the nonlinear analysis results to the .f06 file.
- 2      write the optimization data controlled by P1 and P2 to the .f06 file.
- 4      save the nonlinear analysis results to the .xdb (or .op2) file.
- 8      save the linear response optimization results to the .xdb (or .op2) file.

For example, by default, results from the nonlinear analysis, the optimization data will be written to the .f06 file and result data will be written to .xdb or .op2.

## ESLRCF,user\_rc\_file

Character\*8, must be lower case. Default = blank

ESLRCF, user\_rc\_file allows a user to define a custom RC file for the internally spawned jobs.

**Example:**PARAM,ESLRCF,*myrc*The contents of the *myrc* file are:

```
mem=200m
exe=-local_path/MSCNASTRAN
del=-local_path/SSS
```

The example shows the user-defined RC file, *myrc*, specifies its own memory allocation and its local Nastran executable and local DMAP database.

**ESLTOPCV**

Default = 80

ESLTOPCV specifies an alternate convergence tolerance for an ESLNRO's topology optimization job. An ESLNRO's topology optimization job will be terminated when the percentage of design variables whose changes are less than a given value, CONVDV specified on the DOPTPRM Bulk Data entry is greater than ESLTOPCV.

**ESLUNT2**

Default = 54

ESLUNT2 is a file unit number used by the nonlinear analysis and the linear response optimization runs in an ESLNRO's topology optimization job for the access of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

**ESLUNT1**

Default = 53

ESLUNT1 is a file unit number used by the linear response optimization run in an ESLNRO's topology optimization job for the storage of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

**EST**

Replaced by the ELSUM Case Control command.

**EULBND**

Default = ELEMENT, SOL 700 only

Defines boundary treatment for Euler boundaries.

**Format:**

PARAM,EULBND,option

|          |   |
|----------|---|
| EXTRAPOL | The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character) |
| ELEMENT  | The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element. (Character)                                  |

**Remarks:**

1. The finite volume representation in general assumes that element values are constant within each element. While this assumption is adequate for the large majority of applications, fluid models involving hydrostatic pressure gradients require that the pressure gradient be also recognized to exist within the element. When element-internal hydrostatic gradients are not accounted for, the calculation will be less accurate and will suffer from numerical symptoms like pair forming of element pressures. By activating the option EXTRAPOL hydrostatic gradients inside the element are taken into account. For meshes without bias, option EXTRAPOL only modifies the numerical schemes along the boundary.
2. When coupling surfaces are used DYPARAM,FASTCOUP has to be activated as well.

**EULSTRES**

Default = VOLUME, SOL 700 only

Defines the update logic for stresses when material is transported in Euler elements.

**Format:**

PARAM,EULSTRES,option

|        |   |
|--------|---|
| VOLUME | The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character) |
| MASS   | The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element.  |

**Remarks:**

1. Only used for the MMSTREN solver.
2. Stresses are a material property and when material flows in or out an element the stress state in the element is changed. This is analogous to temperature and energy. Not the temperature is transported, but energy. After transporting energy the temperature is re-computed by dividing the energy by element mass and specific heat. In case of stress the “energy” is given by mass times stress. After transporting this “energy” the new stress follows by dividing it by mass. As shown in Chapter 6 of the Theory Manual this gives a correct updating procedure for stresses. There it also proven that stress times mass is conserved during transport.
3. In most simulations variations in density are small and the multiplication by mass can be replaced by a multiplication by volume. This method is activated by option VOLUME which is the default option. Using the MASS option may have some influence on simulations with large density variations. The option MASS will give the most accurate results.

4. The transport logic of the effective plastic strain is identical to that of stresses. When using option MASS the plastic strain is computed more accurately when material is compressed.
5. The (single material) Euler with strength solver makes use of the multiplication by mass. The multiplication by volume is not implemented for this solver.

## EUSUBCYC

Default = 1, SOL 700 only

Controls the maximum growth of the subcycling interval in Euler computations.

### Example:

PARAM,EUSUBCYC,2

### Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

## EXCLUDE

Default = 0.

PARAM, EXCLUDE, Integer. Add PARAM, EXCLUDE, n in the buckling subcase(s) to activate linear buckling with local exclusion of differential stiffness. If n is positive, it selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be ignored. If n is negative, the absolute value selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be retained.

To select the GRID points care must be exercised by the user which follow natural mesh lines as much as possible, otherwise an element may be connected to some GRID points that have differential stiffness and some GRID points that do not. Sometimes this is impossible to ensure, and so the transition zone between retained-excluded differential stiffness should not be in, or close to, a region of interest. Selecting stiff (unlikely to buckle) zones for the transition zones is also advised.

Multiple buckling subcases with different exclude values are allowed. For example:

```
DISPLACEMENT (PLOT, SORT1, REAL) =ALL
SUBCASE 1
    SUBTITLE=Static Load
    SPC = 2
    LOAD = 2
    analysis=statics
subcase 2
    SUBTITLE = Buckling Subcase
    SPC=2
    statsub=1
    analysis=buck
```

```
METHOD=105
param,exclude,-1
subcase 3
SUBTITLE = Buckling Subcase
SPC=2
analysis=buck
statsub=1
METHOD=105
param,exclude,-2
BEGIN BULK
```

## EXTDR

EXTDR Default = NO. See EXTOUT.

## EXTDROUT

EXTDROUT Default = NO.

## EXTDRUNT

EXTDRUNT Default = 31. See EXTOUT.

## EXTOUT

When inputting the matrices for a reduced external superelement (SEBULK, CSUPER), there are four options that can be selected using the parameter EXTOUT. EXTOUT must be placed in the Case Control Section above any subcase or in the main Bulk Data Section. The options for Step 1 (see the table at the end of this discussion) are as follows:

If EXTOUT is set to MATRIXDB, the reduced structural matrices and loading are stored on the database.

If EXTOUT is set to DMIGDB, the reduced structural matrices and loading are stored on the database in a format which allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model.

If EXTOUT is set to DMIGOP2, the reduced structural matrices and loading are written in OUTPUT2 format to a tape unit specified by the parameter EXTUNIT (Default is 31). The storage format is the same as the DMIGDB option and allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model. The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section.

If EXTOUT is set to DMIGPCH, the reduced structural matrices and loading are output on the punch file (.pch) in DMIG format. Also, the model cannot have rotors (ROTOR, ROTORG, ROTORSE, and ROTORAX Bulk Data entries).

The procedure for accessing the external superelement information depends on the option used to output the external superelement in Step 1. The methods are as follows:

1. If EXTOUT was MATRIXDB or DMIGDB in Step 1, use the following commands in the File Management Section:

ASSIGN SEXXX='step1.MASTER'  
DBLOCATE DATABLK=(EXTDB, EXTROTDB)CONVERT(SEID=xxx),  
LOGICAL=SEXXX

where:

step1.MASTER is the database from the Step 1.

xxx is the superelement identification number given to the partitioned Bulk Data Section for the external superelement.

2. If EXTOUT was DMIGOP2 in Step 1, then use the following commands in the File Management Section:

ASSIGN INPUTT2='step1\_output2\_file',UNIT=extunit

where:

step1\_output2\_file is the OUTPUT2 file from Step 1.

extunit is the unit number specified by the parameter EXTUNIT (Default=30).

3. If EXTOUT was DMIGPCH in Step 1, then include the punch file from step one in the partitioned Bulk Data Section. In addition, add the following Case Control commands in the subcase for the external superelement:

K2GG=KAAX

P2G=PAX

The SEBULK entry defining the superelement as an external superelement and the EXTRN entry in the partitioned Bulk Data Section should not be specified.

If data recovery is desired for the external component in SOLs 101, 103, and 107 through 112, there are three methods to transmit the displacements of the reduced model to the external full model. The method is selected by the parameter EXTDROUT in the partitioned Bulk Data Section. The options are as follows:

1. EXTDROUT set to MATRIXDB. The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model.
2. EXTDROUT set to DMIGDB. The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.
3. EXTDROUT set to DMIGOP2. The same as EXTDROUT set to DMIGDB except that the displacements of the reduced model are written in OUTPUT2 format to a tape unit specified by parameter EXTDRUNT (Default=31). The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.

Data recovery for the external component is limited to SOLs 101 and 103 and 107 through 112 and 400. Data recovery is accomplished using a restart procedure from the data base created in step one and setting parameter EXTDR to YES. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The input methods are as follows:

1. If EXTDROUT was MATRIXDB or DMIGDB in Step 2, then add the following commands in the File Management Section:

```
ASSIGN SEXX='step1.MASTER'  
RESTART LOGICAL=SEXX  
ASSIGN SEYYY='step2.MASTER'  
DBLOCATE DATABLK=(EXTDB) WHERE(SEID=yyy),  
LOGICAL=SEYYY
```

where:

step1.MASTER is the database from the Step 1.

step2.MASTER is the database from the Step 2.

yyy is the superelement identification number given to the partitioned Bulk Data Section for the external superelement in Step 2.

2. If EXTDROUT was DMIGOP2 in Step 2, then add the following commands in the File Management Section:

```
ASSIGN INPUTT2='step2_output2_file',UNIT=extdrunt
```

where:

step2\_output2\_file is the OUTPUT2 file from Step 2.

extdrunt is the unit number specified by the parameter EXTDRUNT (Default=31).

For SOL 101, the Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in step one to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.

For SOL 103 and 107 through 112, the Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, FREQ/DLOAD or TSTEP/DLOAD subcases.

| Step 1 - Create External SE | Step 2 - Perform Analysis   | Step 3 - Data Recovery for External SE  |
|-----------------------------|---|---|
| PARAM,EXTOUT, MATRIXDB      | a. ASSIGN SEXX='step1.MASTER'<br>DBLOCATE DATABLK=(EXTDB, EXTROTDB),<br>CONVERT (SEID=xxx),<br>LOGICAL=SEXX<br><br>b. PARAM,EXTDROUT,MATRIXDB   | a. ASSIGN SEXX='step1.MASTER'<br>RESTART LOGICAL=SEXX<br>ASSIGN SEYYY='step2.MASTER'<br>DBLOCATE DATABLK=(EXTDB),<br>WHERE(SEID=YYY),<br>LOGICAL=SEYYY<br><br>b. PARAM,EXTDR,YES  |
| PARAM,EXTOUT, DMIGDB        | a. ASSIGN SEXX='step1.MASTER'<br>DBLOCATE DATABLK=(EXTDB, XTROTDB),<br>CONVERT (SEID=xxx),<br>LOGICAL=SEXX<br><br>b. PARAM,EXTDROUT,MATRIXDB or<br>PARAM,EXTDROUT,DMIGDB or<br><br>PARAM,EXTDROUT,DMIGOP2 | a. ASSIGN SEXX='step1.MASTER'<br>RESTART LOGICAL=SEXX<br>ASSIGN SEYYY='step2.MASTER'<br>DBLOCATE DATABLK=(EXTDB)<br>WHERE(SEID=YYY)<br>LOGICAL=SEYYY<br><br>ASSIGN INPUTT2= 'step2_output_file',<br>UNIT=Extdrnt<br><br>b. PARAM,EXTDR,YES  |
| PARAM,EXTOUT, DMIGOP2       | a. ASSIGN INPUTT2='step1_output2_file',<br>Unit = extunit<br><br>b. PARAM,EXTDROUT,MATRIXDB or<br>PARAM,EXTDROUT,DMIGDB or<br><br>PARAM,EXTDROUT,DMIGOP2  | a. ASSIGN SEXX='step1.MASTER'<br>RESTART LOGICAL=SEXX<br>ASSIGN SEYYY='step2.MASTER'<br>DBLOCATE DATABLK=(EXTDB),<br>WHERE(SEID=YYY),<br>LOGICAL=SEYYY<br><br>ASSIGN INPUTT2= 'step2_output_file',<br>UNIT=Exdrnt<br><br>b. PARAM,EXTDR,YES |
| PARAM,EXTOUT, DMIGPCH       | Include the .PCH file in partitioned Bulk Data Section<br>and for external SE subcase<br><br>a. K2GG=KAAX<br>PG=PAX   |   |

## EXTRCV

Default = 0

EXTRCV > 0 indicates that data recovery is to be performed on an external superelement. In this type of run, the database for the external superelement must be attached as the primary database ([Database Concepts](#), [453](#) in the *MSC Nastran Reference Guide*), and the database that contains the solution vectors, superelement map, and external/internal grid point equivalence table for its downstream superelement must be attached via the DBLOCATE statements. The value of EXTRCV must also be specified in the CONVERT clause of the DBLOCATE statement for the EMAP data block.

The following example shows the DBLOCATE statements for external superelement data recovery in SOL 101.

## EXTUNIT

Default = 31. See EXTOUT.

## FACTOR

Default = 10000

See OLDSEQ.

## FASTFR

Default = AUTO

In MSC Nastran 2004, the FASTFR method was introduced for modal frequency response analysis. It can be selected via the Bulk Data entry, PARAM,FASTFR,YES and shows significant performance improvement for certain models in the mid-frequency range. PARAM,FASTFR,NO deselects the FASTFR method causing the program to use the standard method for modal frequency analysis.

By default, the program will decide automatically which solution method will be most efficient for the frequency response part in a SOL 111 analysis. Based on the size of the modal space and some other heuristic criteria, either the FASTFR solution method will be run, or the FRRD1 module with or without the iterative solver will be used.

Limitations for the FASTFR method

1. The FASTFR method works only for SOL 111 and for SOL 200 with ANALYSIS=MFREQ.
2. SESDAMP and FASTFR are not allowed in the same run.
3. To force the iterative solver in FRRD1, FASTFR=yes is required.

## FBATOLR

Default = 1.0E-05

This is the tolerance that is applied to grid point coordinates in order to determine connections between potential connection grid points of various FRF components in the FBA process.

The default tolerance should be satisfactory for most situations. A looser tolerance may be needed in certain situations. An example is the case where the potential connection points of an FRF component are associated with the shell elements of RSSCON solid-to-shell element connectors. In this case, a looser tolerance may need to be specified in order to achieve proper connections between FRF components in the FBA process.

## FBLEND

Default = 0.6667, SOL 700 only

Eulerian elements with uncovered fractions smaller than FBLEND are blended with adjacent elements to form a clump so that they do not control the time step.

### Format:

PARAM,FBLEND,VALUE

### Example:

PARAM,FBLEND,0.5

VALUE      The uncovered fraction below which blending occurs. ( $0.0 \leq \text{Real} < 1.0$ )

**Remarks:**

1. The default value is satisfactory for virtually all calculations.
2. Elements are blended only if they would have controlled the time step otherwise.
3. Elements with uncovered fractions greater than FBLEND are not blended and are allowed to control the time step.
4. Large values of FBLEND produce a larger time step but many blends. Small values produce a smaller time step and fewer blends.
5. In a calculation with a coupling surface, STEPFCT is smaller or equal FBLEND to avoid instabilities (see PARAM,STEPFCT).

**FDRLDS**

Default = YES

PARAM, FDRLDS, YES turns on the logic to generate additional residual vectors for SOL 111 based on the differences of structural and structural damping matrices. For free-free models, a SUPPORT entry may improve the free-free modes and yield additional augmented eigenvalues (PARAM, BAILOUT, -1 may occasionally be needed even for a good static SUPPORT set). PARAM, FDRLDS, NO can be used to turn off the generation of frequency dependent residual vectors.

## FIXEDB

Default = 0

FIXEDB is used to reduce the cost of superelement checkout.

|           |  |
|-----------|--|
| -2        | (SOL 101 only) is used on the initial runs when the user suspects that the superelement may contain errors and that only operations necessary for fixed-boundary solutions need be performed. In particular, the generation of the $[G_{oa}]$ matrix is branched over in the SEKR operation and $[P_a]$ is not generated in the SELR operation. These operations typically result in 50% of the reduction cost and are not needed in the fixed-boundary data recovery operations described in the next paragraph. After this operation has been completed, the keyword SELANG will appear in the database dictionary, indicating that the $[P_a]$ stored there is incomplete, and should not be summed into the downstream superelement, because System Fatal Message 4252 will be issued.   |
| $\leq -1$ | (SOLs 101 and 103 only) allows uncoupled solutions for any superelement with conventional output requests. This output may be obtained as soon as the superelement is successfully generated and reduced and does not require that the entire model be assembled. In superelement statics, the solution is the component due to the $\{u_o^o\}$ vector, i.e., a fixed-boundary solution. In superelement modes, the solution is the uncoupled eigenvectors of the component. If PARAM,FIXEDB,-1 is specified in the Bulk Data or in the residual structure subcase, the modes of the residual structure will not be computed. For a printout or plotting of the component mode eigenvectors it is recommended that PARAM,FIXEDB,-1 be specified in the Bulk Data Section or above the subcase level in Case Control. If the modes of the residual structure are desired, then PARAM,FIXEDB,0 should be specified in the residual structure subcase. Exterior degrees-of-freedom listed on SECSETi and SESUP entries are free, and those on SEBSETi degrees-of-freedom are fixed. Data recovery for the residual structure should not be requested for this option. |
| +1        | (SOL 101 only) is used after the superelement has been proven valid. In the SEKR and SELR operations, it provides a branch over all operations already completed in the SEKR and SELR phases and completes the generation of the $[G_{oa}]$ matrix and the boundary stiffness and load matrices. It is also a method to recover the factor of the $[K_{oo}]$ matrix if the run aborted while computing $[G_{oa}]$ .  |

## FKSYMFACT

Default = 0.024

Follower force stiffness is typically unsymmetric, but in some cases making it symmetric may improve convergence in nonlinear problems, whereas in other cases it may hinder it.

FKSYMFACT controls the symmetrization of the follower force stiffness in SOL 106 and SOL 400. If FKSYMFACT = 1.0 is specified, the follower force stiffness  $K_f$  is symmetrized as:

$$K_{fs} = \frac{1}{2}(K_f + K_f^T)$$

and the symmetric part  $K_{fs}$  is used for efficiency. If FKSYMFAC= 0. is specified, the original follower force stiffness  $K_f$  is used. If a value of  $0. < \text{FKSYMFAC} < 1.$  is specified, the non-symmetric part of the follower force stiffness is calculated as:

$$K_{fn} = K_f - K_{fs}$$

and the ratio of unsymmetry:

$$r = \frac{\|K_{fn}\|}{\|K\|}$$

is compared with the user specified value of FKSYMFAC. The norm  $\| . \|$  is the absolute maximum number of the matrix.

If  $r < \text{FKSYMFAC}$ , the symmetric stiffness  $K_{fs}$  is used.

If  $r > \text{FKSYMFAC}$ , the original unsymmetric stiffness  $K_f$  is used.

The default value for FKSYMFAC was determined by a parametric study and for most cases this will make the follower force stiffness symmetric, which will give sufficiently accurate answers. The asymmetry ratio( $r$ ) increases as geometric nonlinearity intensifies so when it becomes greater than FKSYMFAC, the unsymmetric follower force stiffness is used.

It is considered important to retain the full unsymmetric follower force stiffness, set FKSYMFAC to zero.

The parameter FKSYMFAC is applicable to SOL 106 and SOL 400 only, all other solution sequences symmetrize the follower force stiffness. See parameter FOLLOWK for a list of solution sequences which calculate the follower force stiffness.

## FLEXINCR

Default = NO

In SOL 144, a value of YES will cause the TRIM subcases to be ignored. Instead, the TRIM Bulk Data will be used to obtain the set of Mach, Dynamic pressure and symmetry values for Unit Solutions (Flexible Increments). These data can be archived in the aeroelastic database for subsequent reuse. (Flexible Increments are always computed. This param merely avoids the TRIM subcase if these increments are all that is required.)

## FLUIDMP

Replaced by options on the Case Control command [FLSPOUT \(Case\), 376](#).

## FLUIDNE

Default = 500

The Householder method of eigenvalue extraction is more reliable and is automatically selected for the fluid's system modes if the acoustic cavity is defined in a superelement and there exists fluid boundary points. The switch to Householder occurs if the number of estimated fluid modes is less than or equal to the value specified by PARAMeter FLUIDNE.

## FLUIDSE

Default = 0

PARAM,FLUIDSE,seidf specifies a special superelement reserved for fluid elements. Frequency dependent fluid elements must still be in the residual. The newer partitioned superelements are not supported.

## FMULTI

Default = .10, SOL 700 only

Defines the dimension of the multimaterial element array.

### Format:

PARAM,FMULTI,VALUE

### Example:

PARAM,FMULTI,.25

VALUE            The relative amount of multimaterial elements. (0.0 < Real < 1.0)

### Remark:

The multimaterial Eulerian elements use an overflow array in which to store material data. This array can hold FMULTI times the total number of Eulerian elements. In a problem where more than 10% of the elements have more than one material, the default value of FMULTI must be increased.

## FOLLOWK

Default = YES

In SOLs 101,103, 105, 106, 107, 108, 109, 110, 111, 112, 115, 116, and SOL 400, FOLLOWK=YES (Default) requests the inclusion of follower force stiffness in the differential stiffness. FOLLOWK=NO requests that the follower force stiffness not be included. For FOLLOWK=YES in SOLs 101,103, 105, 107, 108, 109, 110, 111, 112, 115, and 116, a separate static subcase is required and the STATSUB command is also required in the eigenvalue subcase. In nonlinear analysis (SOL 106 and 400), the follower force is included if PARAM,LGDISP,1 is specified. FOLLOWK is ignored in SOL 106 and SOL 400 if LGDISP is not specified.

## FRQDEPO

Default = NO

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

## FULLSEDR

Default = NO

In a run with superelements, PARAM,FULLSEDR,YES will merge results (DISPL, STRESS, etc.) from all of superelements into a single result as if the run contained no superelements. This is not supported for BEGIN BULK superelements (parts) unless the element and grid identification numbers are unique across all part superelements and the residual structure.

## FZERO

Default = 1.0

See AUTOSPRT.

## G, GFL

Default = 0.0

G and GFL specify the uniform structural and fluid-damping coefficient in the formulation of dynamics problems. In coupled fluid-structure analysis, G is applied to the structural portion of the model and GFL to the fluid portion of the model. To obtain the value for the parameter G or GFL, multiply the critical damping ratio,  $C/C_o$ , by 2.0. PARAM,G and GFL are not recommended for use in hydroelastic or heat-transfer problems. If PARAM,G (or GFL) is used in transient analysis, PARAM,W3 (or W3FL) must be greater than zero or PARAM,G (or GFL) will be ignored. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*.

In frequency response and complex eigenvalue analyses, the use of G or GFL will cause the stiffness matrix corresponding to the structural or fluid portion of the model to become complex. The user should be aware that this will, in general, require more memory and resources for the calculations.

## GEOMU

Default = 40

See POST=0.

## GPECT

Default = -1

GPECT controls the printout of all elements connected to each grid point. GPECT=+1 requests the printout. In superelement analysis, the list is printed if PARAM,CHECKOUT,YES is specified or the SEMG or SEALL Case Control command selects the superelement. GPECT=-1 suppresses the printout.

## GRADMESH

Default = OFF, SOL 700 only

Glues fine meshes to coarse meshes. See the section on Graded meshes in the user manual for further information.

### Format:

PARAM,GRADMESH,OPTION

**Example:**

```
PARAM,GRADMESH,MINVOL
```

|        |        |   |
|--------|--------|---|
| OPTION | OFF    | Graded mesh gluing is not used (Character)  |
|        | MINVOL | If an element of one mesh is covered by an element of another mesh the element with the largest volume will be inactivated. It will also be removed from the output request for Eulerian archives.          |
|        | ELNUM  | If an element of one mesh is covered by an element of another mesh the element with the smallest element number will be inactivated. It will also be removed from the output request for Eulerian archives. |

**Remarks:**

1. This parameter can be used to build block-structured meshes.
2. All Euler elements have to be either orthogonal or axial symmetric.
3. To get meaningful physical results, the change in mesh size going from one element to the next should not be larger than 1.4 or smaller than 0.7.

**GRAVSET**

Default = 0

A PARAM, GRAVSET, n where n is a Bulk Data SET1 entry, allows the user to apply gravity loading (GRAV) to just a portion of the structure. GRAVSET like GRAV loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the gravity loading is to be applied. GRAVSET can appear in either the Bulk Data or ABOVE or IN the 1st Subcase. A GRAVSET appearing in any other Subcase will be ignored and may cause incorrect results. A GRAVSET appearing in or above the first subcase takes precedence over a GRAVSET appearing in Bulk Data. See also the companion RFORSET entry. If there is no GRAV loading called out, then PARAM, GRAVSET, n needs to be removed from the run

**GRDPNT**

Default = -1

GRDPNT > -1 will cause the grid point weight generator to be executed. The default value (GRDPNT = -1) suppresses the computation and output of this data. GRDPNT specifies the identification number of the grid point to be used as a reference point. If GRDPNT = 0 or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid-related masses and masses on scalar points are ignored. The following weight and balance information is automatically printed following the execution of the grid point weight generator.

- Reference point.
- Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
- Transformation matrix [S] from the basic coordinate system to principal mass axes.
- Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.).

- Inertia matrix I(S) about the center of gravity relative to the principal mass axes. Note: Change the signs of the off-diagonal terms to produce the “inertia tensor.”
- Principal inertias I(Q) about the center of gravity.
- Transformation matrix [Q] between S-axes and Q-axes. The columns of [Q] are the unit direction vectors for the corresponding principal inertias.

In superelement static or geometric nonlinear analysis, GRDPNT > -1 also specifies the grid point to be used in computing resultants, in the basic coordinate system, of external loads and single point constraint forces applied to each superelement. If GRDPNT is not a grid point (including the default value of -1), then the resultants are computed about the origin of the basic coordinate system. In superelement analysis, weights and resultants are computed for each superelement without the effects of its upstream superelements.

For axisymmetric elements, the GRDPNT output may be misleading as the algorithm is designed for 3D elements. For the TRIAX6 element, the mass is for the entire model and the center of gravity and inertias are for the cross section in the x-z plane. For the hyperelastic TRIAX and QUADX elements, the mass is for one radian and the center of gravity and inertias are for the cross section in the x-y plane. For the harmonic TRIAX and QUADX elements, the mass is for the entire model and the center of gravity and inertias are for the cross section in the element coordinate system x-y plane.

## GUSTAERO

Default = 1

If gust loads are to be computed, for example on restart, set GUSTAERO to -1. The default is recommended if no gust loads are to be computed after the flutter analysis.

## GYROAVG

Default = 0

Used to specify one of two formulations for frequency response analysis using the rotor dynamic capability. The default is to determine any frequency-dependent terms for each frequency. This option activates the frequency-dependent looping option. Setting the value < 0 uses an ‘average’ frequency formulation. This option avoids using the frequency-dependent looping and results in a shorter execution time. For this option, PARAM,WR3 and PARAM,WR4 must be specified to include rotor damping.

## HEATCMD

Character\*16, Default=nastran

Name of a command to run SOL 600 thermal contact runs. Nastran first sets up an Marc run to determine the thermal contact conditions which are output in a file named jid.nthcnt. Next, Nastran converts these to standard Nastran thermal elements, and finally spawns a second Nastran job from the primary Nastran job. The command to run the second Nastran job is provided using this parameter. For example, if nast2005t1 is desired, enter CMD=nast2005t1. If the command “Nastran” is desired, either leave the parameter out or enter “nastran”. The MSC Nastran run to be spawned will have the form:

CMD jid.nast.dat rcf=RCF

Where file RCF depends on PARAM,MARHEATM.

**Remarks:**

1. See PARAM,MRSPAWN2 for structural analysis.
2. CMD will be converted to lower case regardless of the case entered.

**HEATSTAT**

Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

TEMP(LOAD) = Heat Transfer Subcase ID

If this default is not acceptable, then in heat transfer subcase add the Case Control word TSTRU=SID and in structures subcase here

TEMP(LOAD) = SID

See the Case Control command, [TSTRU \(Case\), 632](#). PARAM,NESET is no longer used. HEATSTAT not supported for p-elements.

**HFREQ, HFREQFL**

Default = 1.+30

The parameters LFREQ, HFREQ, LFREQFL, and HFREQFL specify the frequency range in cycles per unit time of the modes to be used in the modal formulations. (LFREQ and LFREQFL are the lower limits and HFREQ and HFREQFL are the upper limits.) In coupled fluid-structure analysis, HFREQ and LFREQ are applied to the structural portion of the model and HFREQFL and LFREQFL are applied to fluid portion of the model. The default for HFREQ and HFREQFL will usually include all vectors computed. Related parameters are LMODES and LMODESFL.

**Note:**

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

**HTOCITS**

Default = 20 (SOL 106 only)

HTOCITS sets the maximum allowable iterations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

## HTOCPRT

Default = NO (SOL 106 only)

PARAM,HTOCPRT,YES requests the printout of the final cold shape's grid locations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

## HTOCTOL

Default = 1.E-2 (SOL 106 only)

HTOCTOL is used to determine convergence of cold shape in hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command). The parameter is used to compare the geometries as the model deforms from its "hot" to "cold" shape.

## HTSYM

Default = 0

This parameter controls the decomposition method for SOL 400 thermal analysis.

- = 0 Use symmetric decomposition solver
- = 1 Use unsymmetric decomposition solver

## ICOPT

Default = 1, SOL 400 Only

Parameter ICOPT works together with the NLIC Case Control Command. The user input loads may or may not be in equilibrium with the initial condition. If ICOPT=0, MSC Nastran will compute the initial acceleration based on user's inputs. Otherwise, it will be assumed that the initial acceleration is null. In other words, when ICOPT=1 (the default), it is assumed the whole structure is in equilibrium automatically. Theoretically, ICOPT=0 gives better solution. However, due to that the matrix is highly singular, a large amount CPU time may be required and the accuracy of the result may be in doubt for the solution with ICOPT=0.

## IFP

Default = value of PARAM,DBALL.

See DBALL.

## INREL

Default = 0

INREL controls the calculation of inertia relief or enforced acceleration in linear static analysis and buckling. INREL = -1 or -2 requests that inertia relief or enforced acceleration be performed.

Enforced accelerations, if desired, are input on the DMIG,UACCEL Bulk Data entry. (See Section 7.2 of the *MSC Nastran Reference Guide* for the theoretical basis.)

Inertia Relief is not currently supported with external superelements and in the contact analysis.

- 1 SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. The total number of degrees-of-freedom specified on SUPORT and SUPORT1 entries must be less than or equal to six.  
In SOL 105, SUPORT1, not SUPORT, Bulk Data entries must be used to define the supported degrees-of-freedom and the SUPORT1 Case Control command may only be specified in a separate static subcase.  
Loads due to unit rigid body accelerations at the point referenced by PARAM,GRDPNT are computed and then appended to the external loads. If PARAM,GRDPNT is specified in superelement analysis, then the point must be interior to the residual structure and exterior to all superelements.
- 2 The value of PARAM,INREL,-2 will provide an inertia relief analysis without the use of SUPORTi entries. To use this capability the structure must contain six and only six rigid body degrees of freedom. SUPORTi entries must not be present when using this option. If the structure has either more or less than six rigid body degrees of freedom, the analysis will either fail or give incorrect results.

## INRLM

Replaced by the INRLOD keyword on the RESVEC Case Control command.

## IRES

Default = -1

IRES=1 requests that the residual load vectors RULV and RUOV be output in all solution sequences. In superelement analysis, the parameters PRPA and PRPJ may also be used to request output of the partial load vectors  $\{P_a\}$  and  $\{P_j\}$ , respectively. In geometric nonlinear analysis, PARAM,IRES,1 will cause the printing of the residual vector

$$\{\Delta P_f\} = [K_{ff}] \{u_f^{n+1} - u_f\} + \{F_f\} - \{P_f\}$$

## ITAPE

Default = -1

ITAPE specifies the output status of the DSCMR matrix in SOLs 101, 103, and 105; and the DSCMCOL table and the DSCM2 matrix in SOL 200. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

## IUNIT

Default = 11

IUNIT specifies the FORTRAN unit number on which the DSCMR matrix in Design Sensitivity SOLs 101, 103, and 105 and the DSCMCOL table and the DSCM2 matrix in SOL 200 will be written. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

## JWLDET

Default=NOLINK

With the default multiple denotations with EOSJWL are NOT LINKED: the detonation wave of one explosive cannot ignite another explosive. When the option is LINK, multiple denotations with EOSJWL are LINKED: the detonation wave of one explosive can ignite another explosive. For option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

## KDAMP, KDAMPFL

Default = 1

If KDAMP or KDAMPFL is set to -1, viscous modal damping is entered into the complex stiffness matrix as structural damping. In coupled fluid-structure analysis, KDAMP is applied to the structural portion of the model and KDAMPFL to the fluid portion of the model. See [Superelement Analysis in the MSC Nastran Reference Guide](#).

## KDIAG

Default = -1.0 (SOLs 106, 153, and SOL 400 with non-contact analysis), or 0.0 ( SOL 400 with contact analysis)

In SOLs 106 (nonlinear static analysis), 153 (steady state heat transfer), and SOL 400 (nonlinear static and transient analysis), KDIAG may be used to eliminate spurious mechanisms and singularities in the nonlinear stiffness matrix. The absolute value of KDIAG will be added to some or all of the diagonal terms in the nonlinear stiffness matrix as follows:

- < 0.0      Then add the absolute value of KDIAG to the null diagonal terms only (SOLs 106 and 143).  
For SOL 400, the absolute value of KDIAG is added to the diagonal term of null columns only.
- = 0.0      Then no action is taken.
- > 0.0      Then add the value of KDIAG to all diagonal terms.

## KDMFILT

Default = 0

- |   |   |
|---|---|
| 0 | Differential stiffness correction will NOT be added to rigid body behavior of pre-stressed models for modal analysis. |
| 1 | Differential stiffness correction will be added to rigid body behavior of pre-stressed models for modal analysis.     |

1. Currently, PARAM, KDMFILT, 1 is active in SOL103 with a preload subcase (STATSUB) and a single modal (METHOD) subcase.
2. Currently, PARAM, KDMFILT, 1 is active in SOL400 with a series of preload steps and a single ANALYSIS=MODES step. Also, in SOL400, the correction is not available when enhanced nonlinear elements are used via NLMOPTS or PSHLN1, etc., type entries.

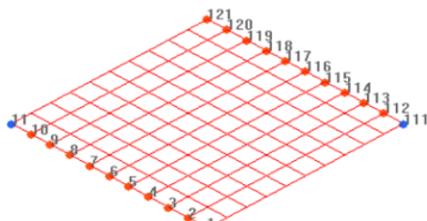
In nonlinear mechanics, in theory a tangent matrix is computed, using a deformation gradient. By polar decomposition, a consistent rotation matrix can be obtained.

In practice, most finite elements such as beam, shell and solid elements, are developed in a heuristic fashion with polynomial type functions representing displacement and volume characteristics. The tangent stiffness is composed of several matrices. A linear K matrix, a  $K_D$  for differential stiffness or geometric stiffness, and often a  $K_F$  for follower load stiffness.

In general, the  $K_D$  matrix which represents a stiffening to the structure due to pre-load of the structure, does not meet rigid body requirements. When a structure is preloaded with a self-equilibrating set of loads and statically supported, for a free-free modal analysis of the pre-loaded structure the static support set is removed. Often because of the deficiency of the  $K_D$  matrix to correctly represent rigid body modes, the resulting free-free modes are poor or missing.

PARAM, KDMFLT, 1 turns on a method to orthogonalize and normalize the computed rigid-body modes.

It cannot be over emphasized the importance of the need to have a self-equilibrating set of loads using a static support system.



The above structure represents a plate structure to be tension pre-stressed in the x-direction. This is accomplished by applying FORCE1 entries on grids 111 thru 120 in the positive x-direction defined by grids 1 thru 11 to grids 111 to 121 and a set of FORCE1 entries along grids 1 thru 11 in the negative x-direction defined by grids 111 thru 121 to grids 1 thru 11.

A static support set is applied at grids 1, 11, and 111. This set is removed for the modal analysis.

If, instead of the FORCE1 entries along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction, a self-equilibrating loading would not be obtained. This is so, because FORCE1 entries are follower loads while SPC1 entries are not. With the SPC1 entries, the structure would not be in true static equilibrium upon removal to the static support set.

If, instead, along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction and non-follower FORCE entries along the grids 111 thru 121, an almost self-equilibrating loading is achieved and PARAM, KDMFLT, 1 will get six good free-free modes. In this case, however, the Lanczos Method has some stability issues and AHOU should be used.

## K4RITZ

Default = 0

Setting PARAM,K4RITZ to a positive value activates logic to generate additional residual vectors for SOL 111 when using ACMS to compute the modal space. Structural damping degrees of freedom (K4) are used to augment the existing set of residual vectors in order to capture damping effects when GE inputs might contribute significantly to overall frequency response. The value of K4RITZ specifically controls the number of iterations performed to compute the basis vectors. Due to extra computation required, this parameter should be used with caution and should rarely be set greater than 1.

## K6ROT

Default = 100.

K6ROT specifies the scaling factor of the penalty stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The contribution of the penalty term to the strain energy functional is

$$\Pi_p = 10^{-6} K6ROT \frac{1}{2} G \int_A (\Theta_z - \Omega_z)^2 t dA \text{ with } \Omega_z = \frac{1}{2} \left( \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right)$$

where  $A$  is the area of the shell element,  $t$  is the shell thickness,  $G$  is the in plane shear modulus, see the MID1 material identification number on the PSHELL Bulk Data entry. The in plane displacements  $u_x, u_y$  and the normal rotation  $\Theta_z$  are shown in [Figure 6-1](#). The normal rotation has no physical meaning and should be ignored. The penalty stiffness removes the singularity in the normal rotation. A higher value than K6ROT=100. is not recommended because unwanted stiffening effects may occur. If you use only the MID1 field, Nastran automatically sets K6ROT=0.0

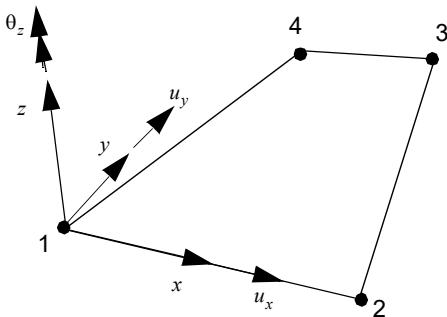


Figure 6-1 In plane displacements  $u_x$ ,  $u_y$  and normal rotation  $\Theta_z$

## LANGLE

Default = 1

LANGLE specifies the method for processing large rotations in nonlinear analysis. By default, large rotations are computed with the gimbal angle method in nonlinear analyses SOLs 106, 129, 153, and 159 with geometric nonlinearity (PARAM,LGDISP,1). If PARAM,LANGLE,2 is specified, then they are computed with the Rotation Vector method. The value of LANGLE cannot be changed in a subsequent restart. For SOL 400, users should not use LANGLE. SOL 400 will use the appropriate method depending on type of element or type of analysis.

## LDSUM

Default = 0

Dictates what trim information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM, XYUNIT, n. LDSUM has the following options:

|   |   |
|---|---|
| 0 | (Default) – Do not create a CSV file for static aeroelasticity  |
| 1 | Create a CSV file that contains for each static aeroelastic subcase: <ul style="list-style-type: none"> <li>■ Subcase ID</li> <li>■ Mach number</li> <li>■ Dynamic Pressure</li> <li>■ Trim Values</li> <li>■ Mass and CG information (mass, xcg,ycg,zcg, IXX,IYY,IZZ,IXY,IXZ and IYZ)</li> </ul> |
| 2 | Same as 1 plus net structural monitor point (MONPNT1, MONDSP1, MONPNT2, MONPNT3) results.   |
| 3 | Same as 2 plus the output of RIGID AIR, ELASTIC RESTRAINED, and INERTIAL, RIGID APPLIED and ELASTIC APPLIED components for the structural MONPNT1 results.  |
| 4 | Same as 3 plus the output of RIGID AIR and ELASTIC RESTRAINED components for aerodynamic MONPNT1 results.   |

## LFREQ, LFREQFL

Default = 0.0

See HFREQ, HFREQFL

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

## LGDISP

LGDISP is a global parameter. For the case with multiple LGDISP definitions, Nastran will use the first definition for all the subcases and steps. Default = -1

- $\geq 0$  The differential stiffness for structural elements is computed for the linear elements and added to the differential stiffness of the nonlinear elements.
- 1 All the nonlinear structural element types that have a large displacement capability in SOLs 106, 129, 153, 159, 600 and SOL 400 (see [Table 3-1](#), under “Geometric Nonlinear” in the *MSC Nastran Reference Guide*) will be assumed to have large displacement effects (updated element coordinates and follower forces). For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).
- 1 No large displacement effects will be considered.
- 2 Follower force for structural elements effects will be ignored but large displacement effects will be considered. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).
- 11 Nonlinear structural elements will be treated the same as LGDISP = 1 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on the current updated geometry (updated element coordinates for the thermal stiffness, element fluxes and external loads).
- 12 Nonlinear structural elements will be treated the same as LGDISP = 2 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on updated current geometry (updated element coordinates for the thermal stiffness and element fluxes, original geometry for external loads).

## LMFACT

LMFACT and PENFN are the scale factor and penalty function for the Lagrange rigid elements and the contact analysis. For Lagrange rigid elements, please see Case Control command, RIGID. The purpose of LMFACT and PENFN is to make the values of stiffness matrix of the Lagrange rigid elements and/or the

contact components about the same relative magnitude as those of the other elements in the model. Too small a value will produce inaccurate results and too large a value will produce numerical difficulties. The same value is usually assigned to both LMFACt and PENFN. Under special requirement, user may assign different values for LMFACt and PENFN. For example, if PENFN=0.0 and LMFACt ≠ 0.0 , then the solution method for the rigid elements becomes the pure Lagrange multiplier method instead of the augmented Lagrangian method. However, user must exercise caution if different values are assigned to LMFACt and PENFN. MSC Nastran will compute the appropriate default values for LMFACt and PENFN. The default value is 1.0e+5 for all solution sequences except SOL 400 and SOL 101 contact.

For SOL 400 and SOL 101 contact, MSC Nastran will compute the appropriate default values for LMFACt and PENFN. These defaults are currently calculated as 0.05% of the average stiffness of the diagonal terms of the stiffness matrix. This computed default is good in general. But when the material/element stiffnesses (e.g., Young's Modulus, Spring Stiffnesses) used in the model vary in a wide range, users may have to adjust the computed LMFACt and PENFN. Note that the computed LMFACt, PENFN are printed out in a .f06 file. It is generally recommended that for these cases, the LMFACt and PENFN should be reduced by several orders of magnitude to avoid numerical difficulty.

If in a coupled multi-physics analysis distinct values for LMFACt and PENFN are desired for each physics pass, the parameter definitions must be made inside the SUBSTEP Case Control command. If no values are entered each physics pass computes its own defaults.

## LMODES, LMODESFL

Default = 0

LMODES and LMODESFL are the number of lowest modes to use in a modal formulation. In coupled fluid-structure analysis, LMODES specifies the lowest modes of the structural portion of the model and LMODESFL the modes of the fluid portion of the model. If LMODES (or LMODESFL) = 0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

In SOL 103, LMODES may be used to reduce the number of eigenvectors to be processed in data recovery which may significantly reduce the CPU and storage costs.

**Note:**

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

## LOADU

Default = -1

See POST=0.

## LOOPID

Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153 the restart proceeds from the last loop ID of the subcase defined by SUBCASID or SUBID. In SOLs 106, and 153 PARAM,SUBID or SUBCASID may also be specified.

## LSTRN

Replaced by the STRAIN Case Control command.

## MACH

Default = 0.0

Mach number. If more than one Mach number was used to compute aerodynamic matrices, the one closest to MACH will be used in dynamic aeroelastic response analysis. The default causes the matrices computed at the lowest MACH number to be used.

## MARALPHA

Default = 10, SOL 600 only.

Determines what type of coefficient of thermal expansion (CTE) is entered in the Nastran input file when CTE varies with temperature. For most Nastran solution sequences, such as SOL 106 and SOL 400, if CTE's vary with temperature, secant values need to be entered. This parameter specifies whether secant or actual values are entered at each temperature on curves using the TABLEM1 entry. In addition, it offers methods to convert the secant CTE's instantaneous values required by Marc. Please read MAT1 Remark 10 and MATHE Remark 8 for further details.

- 1      CTE's (vs temperature) are entered in the Nastran input as instantaneous values – no conversion is required (Warning if this option is used, the CTE's should only be used in SOL 600 and will produce incorrect results if used in SOL 106 or SOL 400).
- 0      CTE's (vs temperature) are secant values (as required by SOL 106, 129 and 400). They will be converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = A_i + \left( \frac{A_i - A_{i-1}}{T_i - T_{i-1}} \right) (T_i - T_{ref})$$

- 1      CTE's (vs temperature) are secant values and are converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = \frac{A_i(T_i - T_{ref}) - A_{i-1}(T_{i-1} - T_{ref})}{T_i - T_{i-1}}$$

- 2 CTE's (vs temperature) are secant values and are converted to instantaneous values using the average of the values computed by options 0 and 1.
- N Same as option 1 except N intermediate points are placed between each original CTE vs temperature value to increase accuracy. N must be 3 or greater. Testing has shown that for CTE's that change appreciably over the temperature range, N should be 10 or larger. The total number of points must not exceed 8000.

**Remarks:**

1.  $\alpha$  is the actual (instantaneous) CTE
2.  $T$  is temperature  $T_{ref}$  is specified on MAT1 and other MAT entries.
3. At  $T(1)\alpha_1 = A_1$
4. The above equations are applied at mid temperatures. The first temperature is retained. Starting with the second temperature  $T_{iN} = 0.5 \cdot (T_i + T_{i-1})$  original. Since the new final temperature will be smaller than the original the user should ensure that the final average temperature is large enough to cover the applied temperature range of the analysis.
5. If the CTE's entered in the Nastran input file are secant values, it is recommended that MARALPHA=N with N set to a value of 20 or larger but less than 8000 divided by the number of points in the largest CTE vs temperature TABLEM1 entry.
6. Only option -1 was available prior to MD Nastran 2010 and MSC Nastran 2008.

**MARAUTOC**

Default = 0, SOL 600 only.

Determines whether NLAUTO entries for SOL 600,129 will override the default or not.

- 0 Do not override the default. (Default)
- 1 NLAUTO initial time step and final time will override the defaults.

**Remark:**

The default for the initial time step is to use the DT value from the entry or the first non-zero time value on the TABLED1 entry divided by 100.0 whichever is smaller. The default for the final time is NDT\*DT from the entry or the last time point from the TABLED1 entry whichever is smaller.

**MARBATCH**

Default = 0, SOL 600 only.

Specifies whether Marc will be spawned from Nastran in the "batch" mode or not.

- 0 Marc will be spawned using batch=no. (Default)
- 1 Marc will be spawned using batch=yes.

**Note:** PARAM,MARBATCH,0 requires PARAM,MARCTEMP,1 (which is the default). This combination of parameters will place the Marc log file in the Nastran log file.

## MARBK105

Default = 1, SOL 600 only.

This parameter controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,105.

- 1 Nonlinear eigenvalues are found. In other words, all loads are placed after Marc's END OPTION and default values are used for CONTROL and AUTOSTEP. This option simulates what happens with SOL 106 or SOL 400.
- 1 Linear eigenvalues are found. In other words, all loads are placed before Marc's END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 105. (Default)

## MARBK106

Default = 1, SOL 600 only.

Controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,106.

- 1 Nonlinear eigenvalues are found. In other words, all loads are placed after Marc's END OPTION and default values are used for CONTROL and AUTO STEP. This option stimulates what happens with SOL 106 or SOL 400.
- 1 Linear eigenvalues are found. In other words, all loads are placed before Marc's END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 106.

## MARC4401

Default = 0, SOL 600 only.

Determines whether recycling due to body to body contact occurs.

- 0 Recycling due to body to body contact can occur
- 1 Recycling due to body to body contact is prevented - will add feature,4401 to the Marc input

## MARC7601

Default = 0, SOL 600 only.

Determines whether large deformation RBE3 element will be used.

- 0      Use small deformation RBE3
- 1      Use large deformation RBE3

**Remarks:**

1. MARC7601=0 sets FEATURE,7601 in Marc.
2. MARC7601=1 will not set FEATURE,7601.

**MARCASUM**

Default is -1 for nonlinear analysis and 1 for linear analysis. SOL 600 only.

Marc's assumed strain formulation is used for plane stress, plane strain and solid elements (Marc types 3, 11 and 7). The assumed strain formulation improves the bending behavior of these elements and uses an enriched set of interpolation functions. Assumed strain should be off for analyses with a significant amount of plasticity. In determining the type of analysis (linear or nonlinear) for defaults of this parameter, the SOL 600, ID Executive statement is used. If ID is 106 or 129, the analysis is considered to be nonlinear and the default is -1. If ID is any other value, the analysis is considered to be linear and the default is 1. For nonlinear analyses without plasticity, this parameter should be turned on for models with solid elements.

- 1      Assumed strain is not used.
- 1      Assumed strain is used.

**MARCAUTO**

Default = leave out parameter, SOL 600 only.

Determines which Marc's increment option is used.

- 1      NLPARM entries will be translated to Marc's AUTO STEP option. If contact is present, the number of steps (NINC) is less than 100, it will be reset to 100. Marc will adaptively reduce the number of steps if possible, however, this option forces the first step to be 1% of the total time. If the first step is too large, experience has shown that convergence problems may result. To start with a different initial time step, see options 999 or -999.
- 1      NLPARM entries will be translated to Marc's AUTO INCREMENT option. If contact is present, the number of steps is automatically set to 100. It has been found that certain difficult contact problems which fail using the AUTO STEP option run successfully using AUTO INCREMENT. This option is not available if the only "loading" is rigid contact or velocity control.
- 2      NLPARM entries will be translated to Marc's AUTO LOAD option with no adjustment in the number of steps. Use of the option is not recommended. This option is not available if the only "loading" is rigid contact or velocity control.

- 999 Marc's AUTO STEP option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.
- 999 Marc's AUTO INCREMENT option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.

See PARAM,MARCITER for a similar option. Do not use both MARCAUTO and MARCITER parameters.

### MARCAZEL

Default = 0, SOL 600 only.

Allows a combination of axisymmetric and plane stress elements for 2D analyses. This analysis technique is sometimes used for approximate turbine disk/blade analysis.

- 0 The combination, if present in the input data will cause a Severe Warning and Marc will not be spawned.
- 1 Combination is allowed and all CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR elements will be considered to be plane stress and mate with CTRIA6 elements.

### MARCBEAM

Default = -1, SOL 600 only.

- 1 All CBEAM elements which reference PBEAML will be mapped to Marc element type 14 for all cross if any CBEAM elements in the model reference MATS1 or MATEP entries. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM (where only A, I, J are available) will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 0 CBEAM will be mapped to Marc element type 14 for all cross sections specified with PBEAML. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 1 All CBEAM elements will be mapped to Marc element type 98 and remain elastic regardless of whether the cross section is specified using PBEAM or PBEAML or whether they reference MATS1 or MATEP.

#### Remark:

If PBMARB6 or PBMMNUM6 beam properties are used, MACRBEAM must be set to zero which automatically activates the new forms of Marc element 98 that can include plasticity.

## MARCBODY

Default=0, SOL 600 only.

Control the logic when the specified value of NBODIES entry on BCPARA bulk data card does not match the actual number of contact bodies in the model.

- 0 Analysis continuing with number of bodies actually in the model
- 1 Analysis continuing with number of bodies specified on NBODIES entry on BCPARA bulk data card. (This may produce exit 13 in Marc)

## MARCBUSH

Default = -1.0 if parameter is not entered, no “small” stiffness will replace zero stiffness terms, SOL 600 only.

Determines whether “small” stiffness values will be used instead of zero for the stiffness values in various directions of CBUSHi elements.

- 1.0 No “small” stiffness terms will replace zero stiffness values in any direction.
- 0.0 Stiffness values of  $0.01 * K_{max}$  will be added for any direction that is zero Marc input (applies to both translational and rotational directions)
- Value The value entered will be used to calculate stiffness=Value\*Kmax to replace any zero stiffness values (applies to both translational and rotational directions)

## MARCCBAR

Default = 0, SOL 600 only.

Specifies whether CBAR will be replaced by CBEAM for SOL 600.

- 0 CBAR is not replaced by CBEAM.
- 1 CBAR is replaced by CBEAM (PBAR is replaced by PBEAM, PBARL is replaced by PBEAML).

**Note:**

Use of this parameter is not usually required but might be beneficial in combination with PARAM,MSPEEDSE,1 to speed up translation of models with a large number of CBAR elements particularly when there are large numbers of PBAR entries or PBARL entries.

## MARCCENT

Default = 0, SOL 600 only.

Controls where the element output is generated.

- |   |   |
|---|---|
| 0 | Element output from Marc will be generated for each integration point.  |
| 1 | Element output from Marc will be generated at the center of each element only. This option saves disk space and computer time, but may not catch the maximum stresses or strains. Because the residual load calculation is not accurate, this should not be used in a nonlinear analysis. |

## MARCCON2

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered is the second value on Marc's CONTACT second entry representing the maximum number of entities to be created for any contact surface. |
|-------|---|

## MARCCON3

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered is the third value on Marc's CONTACT second entry representing the maximum number of nodes that lie on the periphery of any deformable contact surface. |
|-------|---|

## MARCCPY

If MARCCPY is specified, Marc files will be copied to Nastran output files and/or deleted according to the option (0, 1, or 2) shown below.

| MARCCPY Option | Copy Marc Output Files to Nastran Output Files | Delete Marc Input & Output Files |
|----------------|--|----------------------------------|
| 0 (Default)    | No   | No                               |
| 1              | Yes  | Yes                              |
| 2              | Yes  | No                               |
| 3              | No   | Yes                              |

If MARCCPY is 1 or 2, the out and log files will be copied as produced by Marc. If MARCCPY is -1 or -2 the actions as shown above for +1 or +2 will occur, and Marc-type test will be converted to Nastran-type text using an ASCII file named marcfilt.txt which must be located in the same directory where the Nastran input resides or in the same directory where the Nastran executable resides.

The following Marc files are potentially affected by the MARCCPY option:

| Marc Output File            | Nastran Output Copied to            | MARCCPY      |
|-----------------------------|-------------------------------------|--------------|
| name.marc.out               | name.f06                            | 1, 2, -1, -2 |
| name.marc.log               | name.log                            | 1, 2, -1, -2 |
| name.marc.t16               | not copied, will remain if produced |              |
| name.op2, fort.11, or ftn11 | not copied, will remain if produced |              |

## MARCDEF

Default = 2, SOL 600 only.

- 0      SOL 600 default options for Marc will be set to values determined to be best for Nastran-type problems (for MARCDEF=0, Marc's SHELL SECT parameter will be set to 11 if the value of MARCDEF is zero).
- 1      Default values will be set to current Marc standard (Mentat) values.
- 2      Default values will be set to "improved" Marc default values agreed on by the Marc and Nastran development groups.

Default values affect the following Marc input data entries and fields:

| MARCDEF Value | MARC Implicit Entry Type | Field | Value           |
|---------------|--------------------------|-------|-----------------|
| 0             | control                  | 2     | 10              |
| 0             | Auto Step                | 5     | 0.01*max time   |
| 0             | Auto Step                | 8     | 10              |
| 0             | Auto Step                | 10    | 1               |
| 1             | control                  | 2     | 3               |
| 1             | Auto Step                | 5     | 1.0E-5*max time |
| 1             | Auto Step                | 8     | 5               |
| 1             | Auto Step                | 10    | 0               |
| 2             | control                  | 2     | 10              |
| 2             | Auto Step                | 5     | 1.0E-3*max time |
| 2             | Auto Step                | 8     | 5               |
| 2             | Auto Step                | 10    | 1               |

**Note:** For MARCDEF=0, the first three values were found to provide better convergence and the last (auto step 10) allows snap-through solution to converge correctly without having to use arc-length methods. This parameter can be set in the system-wide rc file as well as the user's rc file or the local rc file (same directory as the Nastran input data to provide the selected set of defaults for all runs if so desired. If the parameter is entered in the Nastran input data file, it will override any parameters set in any of the rc files.

## MARCDILT

Default = 0, SOL 600 only.

If omitted, SOL 600 determines the value.

- |   |  |
|---|--|
| 0 | Constant dilatation is not used.   |
| 1 | Constant dilatation formulation is used for solids, axisymmetric, and plane strain elements (advance nonlinear element types 7, 10, 11, 19 and 20) if the model includes any of these element types. For elastic-plastic and creep analysis this formulation is usually too stiff when constant dilatation is not used. MARCDILT=1 and MARCASUM=1 should not both be used. |

## MARCDIS2

Default = Program determines value, SOL 600 only.

- |       |  |
|-------|--|
| Value | If entered, this integer value entered here is the second value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of different lists of distributed loads. |
|-------|--|

## MARCDIS3

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered here is the third value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of elements in any particular distributed loads list. |
|-------|---|

## MARCDIS4

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered here is the fourth value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of nodes with point loads applied. |
|-------|---|

## MARCDMIG,N

Default = 0, SOL 600 only.

If matrices or loads are entered using K2GG, M2GG, B2GG, K2PP, M2PP, B2PP, P2G in the Nastran Case Control Section, they will be translated to Marc as follows depending on the value of N:

- |     |  |
|-----|--|
| N=0 | All DMIG's in the Nastran file (and include files) will be placed in the Marc input file whether used or not.  |
| N>0 | All DMIG's in the Nastran file (and include files) will be placed on a new file named dmigxxxx.dmi where xxxx is the value of N. This new file will be "included" in Marc using a Marc include statement. For example, if N=100 the file name will be dmig100.dmi if N=25765 the file is dmig25765.dmi. N must not exceed a value of 999999. |

**Note:** This parameter is ignored for External Superelements (if the MESUPER Bulk Data entry is present).

## MARCDUPE

SOL 600 only.

Controls whether SOL 600 will check for duplicate entries for most every type of bulk data entry. SOL 600 does not allow duplicate entries, but the portion of IFP that runs prior to spawning Marc does not usually check for duplicate entries.

- 1 Duplicate entries will be checked and exact duplicates are found, the job will fatal out.
- 2 In addition to option 1, entries will be checked and if duplicate ID's (field 2 for most entries or field 3 for loads) are found, the job will fatal out.
- 1 Duplicate entries and ID's will not be checked (this is desirable for certain models)

## MARCDYND

Integer, Default = 0

Determines whether dynamic loads with "advanced table input" will be used in SOL 600.

- 0 Do not use advanced table input for dynamic loads
- 1 Use advanced table input for dynamic loads

If MARCDYND=1, PARAM,MARCTOTD,1 and PARAM,MARCTOTT,1 must also be set.

**MARCEKND**

Default = 0, SOL 600 only.

ID           Selects the type of strain results to be placed in a Nastran op2 file (if a request for an op2 file is made). ID can take the following values:

  MARCEKND=0, Total strains will be processed

  MARCEKND=1, Plastic strains will be processed

  MARCEKND=2, Elastic strains will be processed

For creep analyses, creep strain is output if a request for strain output is made.

**MARCEXIT**

Default = 0, SOL 600 only.

0           If one of the COPYR options on the SOL 600 statement is specified, Nastran will process these options and then a DMAP exit will occur.  
1           The COPYR options will be processes and Nastran will not exit.

**MARCFEAT,N**

SOL 600 only.

If entered will add FEATURE,N to the Marc input file in the Parameters Section.

N           Feature to be added, for example PARAM,MARCFEAT,5102 will generate a heat transfer thermal contact file jid.marc.nthcnt

Only one PARAM,MARCFEAT may be entered.

**MARCFILI**

No Default, SOL 600 only.

Name       Name a file name limited to 8 characters (16 characters if param\* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. For example, if CONTINUE=1 on the SOL 600 statement and PARAM,MARCFIL1,DMIG44 is entered, friction stiffness and possibly damping) matrices are created in DMIG format by Marc and placed on file DMIG44. The various CONTINUE options use the following MARCFILI entries:

| Continue Option | MARCFILi | SOL Executed | K2GG/K2PP |
|-----------------|----------|--------------|-----------|
| 1.              | MARCFIL1 | 107          | K2GG      |
| 2.              | MARCFIL2 | 107          | K2GG      |
| 3.              | MARCFIL3 | 111          | K2PP      |
| 4.              | MARCFIL1 | 112          | K2PP      |
| 5.              | MARCFIL1 | *            | *         |
| 6.              | MARCFIL1 | 110          | K2GG      |
| 7.              | MARCFIL1 | 103          | K2GG      |

### Remarks:

- For most continue options other than brake squeal models, the Marc portion of SOL 600 will produce files with very long names such as jid.marc.conmpc\_0007. These file names are too long for a Nastran parameter field even if PARAM\* is used. The solution is to specify a short name such as PARAM,MARCFIL1,ABCD.TXT  
Inside ABCD.TXT place an include line for the actual file desired, such as  
INCLUDE 'myjob.marc.conmpc\_0007'  
Where myjob would be replaced by the actual JID of the primary Nastran run without the bdf or dat extension.
- If INITCON=4 on the BCPARA entry is specified contact MPC's for each increment will be out on a file named jid.marc.conmpc\_incr (if the increment is 7 it would be jid.marc.conmpc\_0007).
- If the DMIGOUT Bulk Data entry is used, DMIG's will be produced in the form jid.marc\_cglsti\_incr. See the description of DMIGOUT for further details.
- For most cases, the difference between jid.marc.conmpc\_last and jid.marc.conmpc.0001 is not large enough to affect the subsequent analysis significantly. To be sure that the last jid.marc.conmpc\_xxxx is used specify the following  
param,marcfil1,lastt  
Nastran will search for all jid.marc.conmpc\_\* files and choose the one with the largest \_xxxx. If no jid.marc.conmpc\_\* files exist, the job will terminate with an fatal error message.

### MARCFRIC

Default = 0.0, SOL 600 only.

When the Case Control command, BCONTACT = ALL is specified, no other 3D contact data is required in the input file, except that the Coulomb coefficient of friction may be entered using the value of this parameter. Do not enter this entry if contact surfaces are specified in the Bulk Data.

### MARCGAPD, D

Default is U0 of the PGAP, 2840 entry, SOL 600 only.

- D Depending on the value of PARAM,MARCGAPP, enter the gap closure distance for fixed direction gaps or the minimum distance between end points for the true distance gap. If  $d > 0$ , the two end points are never closer than a distance  $|d|$  apart. If  $d < 0$ , the two end points are never farther apart than  $|d|$ .

## MARCGAPN, ID

No Default, SOL 600 only.

- ID ID of gap element for which the immediately following PARAM,MARCGAPP and PARAM,MARCGAPD apply. Unlike most other parameters, several sequences of parameters MARCGAPN, MARCGAPP and MARCGAPD may be entered to specify values for all gap elements. If no MARCGFAPN is entered, the values entered for MARCGAPP and MARCGAPD will be used for all gaps in the model.

## MARCGAPP

Default = 0, SOL 600 only.

- 0 Nastran gap elements will be translated to Marc fixed gap elements.
- 1 Nastran gap elements will be translated to Marc True Distance gaps.

## MARCGAUS

Default if parameter is not entered = 1, SOL 600 only.

- 1 SOL 600 output stresses and strains will be at the Gauss points for solid elements and extrapolated to the corner points for plates/shells elements. Strains are handled the same way as stresses.
- 2 SOL 600 output stresses and strains will be at the center and at the grid points for solid elements (the maximum stress from any Gauss point is determined and compatible stresses for that Gauss point are placed at the center and at each of the grid points). Shell/plate stresses are at the center (top and bottom surfaces). The maximum Gauss point stress at each surface is found and a compatible set of stresses at that Gauss point are placed in the center of the surface. Strains are handled the same way as stresses.
- 3 Solid stresses/and strains are the same as option 1 and shell/plate stresses and strains are the same as option 2.

If this parameter is entered with values of zero or less or values greater than 3, it will be reset to 1.

## MARCGLUE

Default = 0 if parameter is not entered, SOL 600 only.

If MARCGLUE is set to 1, all contact surfaces will be glued whether or not IGLUE=1 is specified on the BCTABLE entry or not.

- 0 IGLUE on BCTABLE entries specifies whether or not glued contact is used
- 1 IGLUE on BCTABLE entries will be ignored and glued contact will be used for all contact surfaces.

**Remark:**

For this parameter to function BCTABLE entries must be entered (do not set BCONTACT=ALL).

**MARCGRAV**

Integer, Default = 0

Determines whether PARAM,MNASTLDS,777 will automatically be set if multiple GRAV entries are present in any subcase. This is determined in a simplified fashion by comparing the number of GRAV entries to the number of subcases to save computer time.

- 0 Set PARAM,MNASTLDS,777 for multiple GRAV entries.
- 1 Do not set PARAM,MNASTLDS,777 regardless of the number of GRAV and SUBCASE entries.

**MARCHOST**

No Default, SOL 600 only.

Determines the name of a hostfile to be used with SOL 600 parallel runs. If this parameter is missing, no host file is used and the parallel run will run on one machine. That machine may have several processors and as many processors as specified on the PARAMARC Bulk Data entry will be used. If PARAM,MARCHOST,Name is specified, the hostfile must be generated by the user in a format acceptable to Marc (see the *Marc and Marc Mentat Installation and Operations Guide*). Each line of the hostfile normally lists how many processors are used on each machine. If PARAM\*,MARCHOST is entered, the name is limited to 16 characters (all lower case).

**MARCIAMN**

Default = 1, SOL 600 only.

- 0 Nastran is directed to spawn Marc (as specified by the SOL 600 Executive Control statement or PARAM,MARCRUN), using a full version of Marc. Standard Marc licensing is required.
- 1 A special version of Marc is spawned by Nastran. This version of Marc may have certain features that are not available in the full version. Marc will be spawned from Nastran with the additional command line switch - iam nanl. The licensing for both Nastran and Marc reflect this situation. This option applies only to Marc version 2003 or later. If PARAM,MARCVERS points to a Marc version earlier than 2003, MARCIAMN will be set to zero and a full version of Marc is required.

The parameter may be set in the system-wide rc, the user rc file or as an environmental variable using NASM\_IAMN=0 or 1 (similar to the way values on the SOL 600 statement are set).

## MARCINTC

Default = 2.

Option to ignore or fatal SOL 600 if any CINTC Bulk Data entries are found. This option will also ignore or issue a warning for GMBNDC entries.

- 0 Fatal job if any CINTC entries are found (also issue warning messages if any GMBNDC entries are found).
- 1 Ignore all CINTC and GMBNDC entries in SOL 600.
- 2 Generate the MPC's for CINTC/GMBNDC and combine them with standard MCP entries (if they exist), then use them in the SOL 600 analysis (SOL 600 only).

### Remark:

MARCINTC=2 spawns a secondary SOL 100 Nastran job to generate MPC's for the CINTC's. Licenses from the primary job are released, then reclaimed after the secondary job terminates. The secondary job stops prior to decomp and uses the standard SOL 101 licenses.

## MARCINTF

Default = 1.0D-6.

Threshold value below which MPC coefficients generated by CINTC/GMBNDC are not considered. This parameter is ignored unless PARAM,MARCINTC,2 has been entered.

## MARCITER

Default = 0, SOL 600 only.

Used to control fixed time stepping in SOL 600.

- 0 Fixed time steps or auto time steps will be controlled by PARAM,MARCAUTO.
- N Allows fixed time steps to be used without needing to set the maximum and minimum times to nearly the same value (using Marc's AUTO STEP option). This parameter triggers true fixed time stepping with the other advantages AUTO STEP has over methods such as AUTO LOAD. For example, it uses better numerical damping. If this parameter is entered with a positive integer (N), a value of 2 is placed in the AUTO STEP field 9 and N is placed in field 7.
- 1 This option is similar to PARAM,MARCITER,N (fixed time stepping will be used) except that the time comes from the NLPARAM or entry. This option is not available if the only "loading" is rigid contact or velocity control. See PARAM,MARCAUTO for a similar option. Do not use both MARCAUTO and MARCITER parameters.

## MARCL001

Default = -1, SOL 600 only.

Determines whether Marc's POINT LOAD (without tables) 2<sup>nd</sup> datablock, 3<sup>rd</sup> field will be honored or not. If this value is set to 1 multiple loads at the same dof in the same subcase will usually be summed (however, see item d.).

- 1            The 2<sup>nd</sup> datablock 3<sup>rd</sup> field will be set to 1 (see Remark)
- 1            The 2<sup>nd</sup> datablock 3<sup>rd</sup> field will not be set to 1. If it is necessary to sum the loads, this must be accomplished in one of several ways:
  - a.       Summed in the GUI or other method of data input
  - b.       By setting PARAM,MNASTLDS,1
  - c.       By setting PARAM,MARCTOTT,1
  - d.       For certain simple multiple force/moment cases, SOL 600 can sum them and PARAM,MNASTLDS,1 must be specified to prevent loads from being specified multiple times. However, it may be difficult to determine whether the loads are complex or not, so the use of either option c or b is recommended.

### Remarks:

1. For releases prior to MD Nastran 2010 and MSC Nastran 2009, this field was set to 1 to automatically sum any forces or moments that might have been entered more than once by the GUI (see PARAM,MNASTLDS for a discussion of ways this can happen). This field, when set to one, allows loads entered more than once at the same grid ID in the same subcase to be summed. However, experience shows that this field does not work in certain circumstances, particularly when "total loads" using PARAM,MARCTOTL are requested. PARAM,NASTLDS has been added beginning with MD Nastran 2010 and MSC Nastran 2009 to handle the cases where the 2<sup>nd</sup> datablock 3<sup>rd</sup> field does not work as expected.
2. To maintain backward compatibility with previous runs, set the following parameters:  
PARAM,MARCL001,1  
PARAM,MNASTLDS,0  
PARAM,MARCTOTT,0  
PARAM,MARCTOTL,0
3. If PARAM,MARCTOTL,1 is entered, PARAM,MARCL001,1 will be set unless PARAM,MARCL001,-1 is entered.
4. This parameter may be set in RC files.

## MARCLOWE

Default = 0, SOL 600 only.

Used in conjunction with superelement matrices created by Marc.

- 0 Standard modulus values for all materials will be used.
- 1 All modulus of elasticity values will be changed to 1.0E-9 for the second Nastran run (when Nastran spawns another Nastran run using the SOL 600 continue option. This option is sometimes necessary for cases where Marc creates a superelement or substructure stiffness matrix but does not create a mass matrix. In this case, the second Nastran run will create the mass matrix using standard elements, density and other concentrated and distributed masses but the stiffness created by Nastran will be very low. Essentially the entire stiffness of the model will come from the stiffness matrices created by Marc.

## MARCLUMP

Default = 0, SOL 600 only.

- 0 Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable). (Default)
- 1 Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable).
- 2 Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.
- 3 Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.

### **Remark:**

This parameter is only used with dynamic analysis.

## MARCMAT2

Default = -1.0, SOL 600 only.

Used if g33 = 0.0 on MAT2 entries. Marc will diverge if g33 = 0.0 for MAT2 entries. If the value entered is positive, the value is a multiplier of g11 and g22 to calculate g33 as follows:

$$g_{33} = \text{marcmat2}^*(g_{11} + g_{22})$$

## MARCMAT3

Default = 1, SOL 600 only.

Used if Nastran has generated MAT2 from PCOMP and the MID of MAT2 is 30000001 is greater corresponding to MID3 for PSHELL.

If the value entered is 0, this entry is ignored regardless of the MAT2 MID value.

If the value entered is 1 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all Cij are zero except the following:

C55=g11  
 C56=g12  
 C66=g22

If the value entered is 2 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C<sub>ij</sub> are zero except the following:

C44=g11  
 C45=g12  
 C55=g22  
 C66=g22

If the value entered is 11 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C<sub>ij</sub> are zero except the following:

C55=g11  
 C56=g12  
 C66=g22

If the value entered is 12 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C<sub>ij</sub> are zero except the following:

C44=g11  
 C45=g12  
 C55=g22  
 C66=g22

**Note:**

This entry is ignored unless the MAT2 MID is greater than 30000000.

## MARCMATT

Default = -1 if parameter is not entered, SOL 600 only.

Determines if Marc input file will be created with materials using the table-driven formats or not.

- 1      Table-driven formats for materials will not be used. (Default)
- 1      Table-driven formats for materials will be used

**Remark:**

This parameter can be set in RC files.

## MARCMEM, Value

Default = Program determines value, SOL 600 only.

- |       |  |
|-------|--|
| Value | If entered, the integer value entered here is the second field on Marc's SIZING entry (MAXALL) and is the main memory specification for memory in Marc. This value is entered in MW (the program multiplies it by 1,000,000). For example, if a value of 350 is entered, the number of 350000000 will be placed in the second field of the SIZING entry. |
|-------|--|

## MARCMID3

Default = 0, SOL 600 only.

Controls whether MID3 will be set to the same value as MID2 when the Marc PSHELL option is used (designated by PARAM,MRPSHELL,1 or when the SMEAR option is used on the SOL 600 Executive Control statement.)

- 0      MID3 will not be changed (if zero or blank, it will remain zero or blank). (Default)
- 1      MID3 will be set to MID2. This improves the singularity ratio in some problems without appreciably changing the results particularly when orthotropic material properties are used. It is not necessary if PARAM,MMAT2ANI,11 or a similar option is used to specify anisotropic material properties.

## MARCMNF

Default = 0, SOL 600 only.

Controls creation of an Adams MNF file by Marc for eigenvalue analysis when Marc is spawned from SOL 600.

- 0      MNF file will not be created. (Default)
- 1      MNF file will be created (for Marc 2003, the MNF file is located in the .t19 file, so PARAM,MARCT19,1 must also be specified).

**Note:** It is not necessary to use this parameter if the MDMIOUT Bulk Data entry is used to request an MSC Adams MNF file.

## MARCMPCC

Default = 0, SOL 600 only.

Determines whether SOL 600 converts MPCs to stiff beams or not.

- 0      MPC's are not converted to stiff beams. (Default)
- 1      MPC's are converted to stiff beams.

**Note:** PARAM,MARCMPCC,1 should not be used if RBE's are converted to MPC's using PARAM,MARCRBE2 or PARAM,MARCRBE3.

## MARCMPII

Integer, Default = 2 for DDM parallel executions, SOL 600 only.

Determines whether the MPI service on PC systems will remain running after the job finishes.

- 1      The MPI service will remain running after SOL 600 finishes
- 1      The Intel MPI service, ismpd.exe will be uninstalled before SOL 600 terminates.
- 2      The Intel MPI service, ismpd.exe will be started before each Marc execution and terminated at the end of each Marc execution from within the Nastran portion of SOL 600.
- 771     MSMPI rather than Intel MPI is requested (starting and/or stopping of the ismpd.exe Intel MPI service is ignored)

### Remarks:

1. If SOL 600 terminates abnormally the service may remain running regardless of the value of this parameter.
2. Leaving the MPI service running will speed up the execution of multiple jobs.
3. This parameter may be entered in RC files.
4. Although ismpd.exe is started by the Marc script, it sometimes fails to start properly unless started outside the script. The user can either do this in his own script or set MARCMPII=2 to have Nastran do it. Either of these options improves reliability of the Intel MPI starting.

## MARCND99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all nodal quantities will be generated or not. If MARCND99=1, all Case Control nodal output requests must have the PLOT qualifier or the job may fail.

- 1      A set named ND999999 will be generated to output all nodes for at least one type of nodal output. This is the default of all Case Control nodal requests do not have (plot).
- 1      The set will not be generated. If all nodal Case Control requests have (plot) such as DISP(PLOT)=ALL, ACCEL(PLOT)=ALL, etc. the default is 1 even if the parameter is not entered.

## MARCNOER

Default = 0, SOL 600 only.

Determines action to take when unsupported features are encountered.

- 0      The internal Marc translator will stop and generate FATAL ERRORS if unsupported features in Marc or in the internal translator are encountered.
- 1      If unsupported features are encountered, they are ignored, no FATAL ERROR messages are issued and if requested, Marc will be executed.

## MARCOFFT

Default = -1, SOL 600 only.

Controls whether Nodal Temperatures are specified at the original or offset grid point or both grid points for beams and shells with offsets or connected by RBE2's (see note 2) when PARAM,MAROFSET is set to 0,2,3. For the default MAROFSET=1 where Marc handles the offsets and RBE2's are not added, this parameter is not applicable and will be reset to -1 internally. Applies only if MAROFSET=0.

- 0      Temperatures are applied both at the original grid point and at the offset grid point.
- 1      Temperatures are applied at the original grid point only.
- 2      Temperatures are applied at the offset grid point only.
- 1     Temperature loading is not altered in any way from the Nastran input.

**Note:**

1. Processing time can increase significantly if beam or shell offsets are present and param,marcofft is zero or greater unless PARAM,MOFFCORE,1 is used.
2. If nodes are connected by RBE2's and a temperature is applied at an independent node, the temperature will be applied at the independent node and the first dependent node of the rbe2 if MARCOFFT=0.
3. If MAROFSET is 0, 2 or 3 and a node has applied temperature and is also a part of a standard RBE2 with many grids, the job may abort or the results can be wrong.

## MARCONTF

No Default, SOL 600 only.

- |      |  |
|------|--|
| Name | Name of a file name limited to 8 characters (16 characters if param* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. If entered this file will be used as the input file for the second Nastran execution (after Marc has finished). If specified, this file will be used instead of automatically creating a file named jid.nast.dat from the original jid.dat input. This option allows more versatility in achieving exactly what is desired in the Nastran continuation run input at the expense of additional input data preparation. |
|------|--|

## MARCOOCC

Default = 0, SOL 600 only.

- 0      Standard Marc memory management will be used. If the memory is sufficient the decomposition will be in core. If the memory requirements are too large, an out of core solution will take place.
- 1      An out of core solution will be forced if solvers 0, 2, 4 or 8 are used. This option triggers Marc parameter, OOC (without any other characters).

- 2 An out of core solution will be forced if solver 0, 2, 4 or 8 are used - available starting with Marc 2005 r2. This option generates Marc parameter OOC,0,1.
- 3 Same as option2 except Marc parameter version,4902 will be added. This option triggers Marc parameter OOC,0,1 plus version,4902.

## MARCOPT

Default = 9, SOL 600 only.

Determines which bandwidth optimizer is to be used.

- 0 Marc does not optimize the bandwidth.
- 2 Cuthill-McKee bandwidth optimization is used
- 5 External user-supplied bandwidth optimization is used
- 9 Sloan bandwidth optimization is used. (Default)
- 10 Minimum degree bandwidth optimization is used (only available with the sparse solver)
- 11 Metis nested dissection algorithm (only available with multifrontal direct sparse solver)
- 9999 Set MARCOPT to -9999 if the OPTIMIZE entry is not wanted in the Marc file for example with use by the iterative solver.

## MARCOSET

Default = 0, SOL 600 only.

Determines whether SOL 600 set names will be standard sets or “open sets” for nodes and elements. The standard Marc sets are:

```
DEFINE, ELEMENT, SET, PR00001
DEFINE, NODE, SET, ND001
```

The Marc open sets (OSET) are:

```
DEFINE, ELEMENT, OSET, PR00001
DEFINE, NODE, OSET, ND001
```

- 1 Standard sets are defined.
- 0 Standard sets are defined for small models and open sets are defined for large models.
- 1 Open sets are defined.

**Note:** For Parallel (DDM) analyses, it is sometimes necessary to set MARCOSET=1.

## MARCOTIM

Default = 0, (SOL 600 only and is mapped to Marc's POST 2nd line 9th field.

Determines if Marc is to be processed at selective or at all output times.

- 0 or 1      Marc output data will be processed by Nastran at all converged output times. This option is similar to INTOUT=YES on the NLPARM entry.
- 2            Marc output data will be processed by Marc only at times near 1.0, 2.0, 3.0, etc. For this option, no additional output times will be available in the Marc.t16 and/or.t19 output files. This option is similar to INTOUT=NO on the NLPARM entry.
- N            Marc output data will be processed by Marc every Nth increment for the.t16 and/or.t19 output files. (N > 2)

## MARCOUTR

Default = 1 if Marc single file parallel input is used, Default = 0 if Marc multiple file inputs are used, see Note, SOL 600 only.

Determines how Marc t16 file results will be handled for SOL 600 parallel processing.

- 0            Multiple t16 files, one for each domain will be produced.
- 1            A single t16 file will be produced by Marc. This option requires Marc 2005 or later versions and the parallel run made using the "single file" input (PARAMARC KIND=0). (Default)

**Note:** Whether single file or multiple Marc inputs are used for parallel processing is determined by the PARAMARC Bulk Data entry.

## MARCPARR

Default = 0, SOL 600 only.

Controls options for splitting an Marc file into parts for DDM.

- 0            All Marc files will be created during this run. (Default)
- 1            Nastran will be stopped after the single-processor file has been created and before DDM files are created. If desired, all files may be copied to a backup directory for use with MARCPARR=2.
- 2            The parallel files will be created starting with the single processor file created using the MARCPARR=1 option.
- 3            Same as MARCPARR=2 except the debug option MARCBUG=1 is turned on.

## MARCPENT

Default = 0, SOL 600 only.

Option to specify if CPENTA is mapped to Marc brick element with degenerate nodes or actual penta elements (see Remarks).

- 0 Map CPENTA to Marc brick elements with degenerate nodes. (Default)
- 1 Marc CPENTA to actual Marc penta elements.

### Remarks:

1. Option zero is the default to maintain backwards compatibility.
2. The t16 to op2 version is not available for option 1. That means no op2, xdb, f06 or punch file is available for option 1. Postprocessing must occur using the t16 or t19 file if option 1 is used.
3. Speed options such as invoked using param,mspeedse are available using option 1 only.

## MARCPINN

Default = 0, SOL 600 only.

- 0 Pin flags will be included by created new nodes and appropriate MPC's by the translator in Nastran. (Default)
- 1 Pin flags will be ignored and the translator will continue.
- 2 A severe warning will be issued and Marc will not run.
- 3 The new Marc PIN CODE option will be used. feature,6901 will also be used so that extra nodes are not introduced.

### Remark:

This parameter can be set in RC files.

## MARCPLAS, n

Default = 3 if there is plasticity in the model. SOL 600 only.

This parameter effects the value of Marc's PLASTICITY parameter. The value of n can range from 0 to 6 (0 is the same as 3).

- 1 Additive decomposition using the mean normal method; small strain formulation
- 2 Additive decomposition using the radial return method; small strain formulation.
- 3 Additive decomposition using the mean normal method; large strain formulation using the updated Lagrange procedure.
- 4 Additive decomposition using the radial return method; large strain formulation during the updated Lagrange procedure.
- 5 Multiplicative decomposition (FeFp) using the radial return method and the three field variational principle; large strain formulation using the updated Lagrange procedure.

- 6 Advance nonlinear elements type 3 and 26 (plane stress), 18 and 30 (membrane) using multiplicative decomposition with the radial return method; large strain formulation using the updated Lagrange procedure.
- 1 Ensures that the Marc PLASTICITY parameter will not be used.

## MARCPOS

SOL 600 only.

Determines whether to terminate Marc if a non-positive definite matrix is attempted.

- 0 The run will terminate if a non-positive definite matrix decomposition is encountered.
- 1 Non-positive definite matrices will be solved.

## MARCPOST

Default, SOL 600 only.

Determines the format for the .t16 and .t19 files.

- 1 The .t16 and .t19 files will be created using the Marc default for the version of Marc that is executed.
- 0 Same as MARCPOST=9.
- 1 The .t16 and .t19 files will be created using Marc K2 formats.
- 3 The .t16 and .t19 files will be created using Marc K3 formats.
- 4 The .t16 and .t19 files will be created using Marc K4 formats.
- 5 The .t16 and .t19 files will be created using Marc K5 formats.
- 6 The .t16 and .t19 files will be created using Marc K6 formats.
- 7 The .t16 and .t19 files will be created using Marc K7 formats.
- 8 The .t16 and .t19 files will be created using Marc K8 formats.
- 9 The .t16 and .t19 files will be created using Marc 2000 formats.
- 10 The .t16 and .t19 files will be created using Marc 2001 formats.
- 11 The .t16 and .t19 files will be created using Marc 2003 formats.
- 12 The .t16 and .t19 files will be created using Marc 2005 formats.
- 13 The .t16 and .t19 files will be created using Marc 2005 r3 formats.

**Note:**

It is suggested that a small test case be executed and tested with your postprocessor to determine what version is necessary for your postprocessor.

## MARCP99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all elemental quantities will be generated or not. If MARCP99=1, all Case Control element output requests must have the PLOT qualifier or the job may fail.

- 1 A set named PR999999 will be generated to output all elements for at least one type of element output. This is the default if all elemental Case Control requests do not have (plot).
- 1 The set will not be generated. If all elemental Case Control requests have (plot) such as STRESS(PLOT)=ALL, STRAIN(PLOT)=ALL, etc. The default is 1 even if the parameter is not entered.

## MARCPRN

SOL 600 only.

Controls the amount of contact information printed.

- 0 Detailed contact information is not printed.
- 1 Detailed contact information is printed (this is equivalent to Marc parameter PRINT,2,8).
- 2 Somewhat less detailed contact information is printed (this is equivalent to Marc parameter PRINT,2). Print constraint matrices associated with MPC's, RBAR, RBE2, RBE3 and the formable to deformable contact.
- 5 Marc print option. PRINT,5 will be used. Prints messages when changes in contact status occur.
- 25 Marc print options. PRINT,2,5 will be used.
- 258 Marc print options. PRINT,2,5,8 will be used. In addition to 2 and 5, also prints the displacement and reaction forces in the local coordinate system associated with formable to rigid contact.

For other print options, use the MARCIN Bulk Data entry.

## MARCPRNG

Default = 0, SOL 600 only.

Determines whether geometry is printed in the Marc .out file.

- 0 Most geometry printing is suppressed.
- 1 All geometry is printed.

## MARCPRNR

Default = 0, SOL 600 only.

Determines whether nodal stress and strain output is printed in the Marc .out file.

- 0 Nodal, stress and strain printing is suppressed.
- 1 Nodal, stress and strain printing will occur if Case Control options request it specifically or by default.

## MARCPROG

No Default, SOL 600 only.

- prg “prg” is the name of a program to be executed instead of Marc. The program may be any program desired and must already be fully compiled on the computer system being run. prg is limited to 8 characters, however 16 characters can be used if PARAM\* is entered. “prg” must be a compiled program, not a script or batch file. The name of the program must be in lower case (if not, it will be converted to lower case).

## MARCRACC

Default = 0, SOL 600 only.

SOL 600 does not normally support RACC on the RFORCE entry. The default is to fatal a job when RACC is zero or blank. This parameter may be used to set it to zero or to obtain approximate results.

- 0 If MARCRACC is zero and RACC is nonzero, the job will terminate with an appropriate message (Default).
- 1 If MARCRACC is one and RACC is nonzero, RACC will be set to zero internally and the job will continue.
- 2 If MARCRACC is two and RACC is nonzero, the following will occur:  
for brake squeal, coriolis loading and values (C1, C2, C3) will be placed in Marc ROTATION A 3rd datablock fields 4-6 as follows:  

$$RR=\sqrt{R1^{**2}+R2^{**2}+R3^{**2}}$$

$$C1=RACC*R1/RR$$

$$C2=RACC*R3/RR$$

$$C3=RACC*R3/RR$$

Where RACC, R1, R2, R3 are described on the RFORCE entry.

For standard structural analysis if MARCRACC=2, the value of A in RFORCE field 5 will be altered using the following equation:

$$A_{\text{new}} = \sqrt{A^{**2} + RACC}$$

Marc's ROTATION A entry will be the same as if RACC=0.0 was entered unless A is zero, in which case the third line will have all zero entries. Results should be considered as an approximation.

### Remarks:

1. See PARAM,MARCRCID for related SOL 600 RFORCE uses.

2. MARCRACC=2 only applies for brake squeal and only if the BRKSQ entry is used for releases prior to MD Nastran R2.1 and MSC Nastran 2007.

## MARCRBAL

Default = 0, SOL 600 only.

This parameter is used only for eigenvalue analysis in the Nastran-Marc interface where natural frequencies or buckling modes need to be calculated using the deformed geometry from a nonlinear analysis. The parameter is only necessary if the last nonlinear increment created a non-positive definite matrix. When MARCRBAL=1 is set, the system will be rebalanced and a positive-definite matrix is assured. Do not use this parameter unless it is known that a non-positive definite system occurs just prior to eigenvalue analysis.

## MARCRBAR

Default = 0, SOL 600 only.

Determines how RBAR is treated.

- |   |  |
|---|--|
| 0 | RBARs will be simulated using Marc's Servo Link. This option is best only for small incremental deformation and rotations. (Default)                     |
| 1 | RBARs will be converted to RBE2 with GN=GA, CM=123456, GM1=GB. The MARCRBE2 option specified will then be used to translate the converted RBARs to Marc. |

## MARCRBE2

Default = 3, SOL 600 only.

- |    |  |
|----|--|
| ID | Determines the type of RBE2 used. The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements.   |
| 0  | RBE2s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations.  |
| 1  | RBE2s will be simulated using Marc's TYING type 80 for translation and Servo Link for rotations. This option is capable of larger incremental deformations but requires small rotational increments. The MARCRBE2=1 option is only available if all 6 DOF's are specified in the CM field (4) of the RBE2 entry. |
| 2  | RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 2D analysis.  |
| 3  | RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 3D analysis.  |

**Note:** If RBAR, RROD or RTRPLT elements are found in the model and if MARCRBE2=2 or 3, these elements will be converted to equivalent RBE2's and used with the new Marc RBE2 element during the Marc execution (thus providing higher accuracy for large deformations and/or rotations).

The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements.

## MARCRBE3

Default = 3 SOL 600 only.

Determines the type of RBE3 used.

- 0      RBE3s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations (see option 4 for a similar alternative).
- 2      RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 2D analysis.
- 3      RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 3D analysis. (Default)
- 4      Same as MARCRBE3=0 except that all MPC's due to RBE3 will be placed ahead of all other MPC's. This option might improve the Marc solution for versions where Marc has not implemented AUTOMSET logic (those versions prior to MSC Nastran 2005 r3).

## MARCREVR

Default = 0, SOL 600 only.

Specifies that all rigid surfaces need to be modified.

- 0      Rigid contact surfaces are correct as entered and no changes are made by the translator. (Default)
- 1      All rigid surfaces are entered backwards and will be reversed.

## MARCREVRX

SOL 600 only.

Determines whether coordinates for NURBS2D (BCBODY) will be reversed or not if CQUADX, CTRIA6, and/or CTRIAX6 elements exist.

- 0      Reverse the coordinates (Y becomes X, X becomes Y).
- 1      In addition to reversing the coordinates (option 0) all points are also reversed. For example, if there are 4 points (1,2,3,4) they are reversed to (4,3,2,1). (Default)

- 1      Do not reverse the coordinates.
- 2      Reverse the coordinates for CQUADX and CTRIAZ abut not for CTRIAZ6.

## MARCRIGD

Default = 0, SOL 600 only.

The parameter should only be entered if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 0      All RBEi will not be converted to stiff beams or plates. (Default)
- 1      All RBEi will be converted to stiff beams or plates for the Marc input file using a stiffness scale factor from parameter MARCRSCL. This parameter allows such elements to have large rotations for versions of Marc which do not include large rotations of rigid elements.

### Remarks:

1. This option may not be used if RBAR, RROD or RTRPLT elements are in the model. See PARAM,MSTFBEAM for an alternative.
2. See PARAM,MARCSCLR to specify a scale factor for the “default” properties of these stiff beams.

## MARCSAME

Default = 0, SOL 600 only.

Determines whether SOL 600 runs with multiple subcase having the same LOAD ID or loads (see note below) in more than one subcase will be processed or not. SOL 600 will usually run under such circumstances but may get the wrong results.

|                   |   |
|-------------------|---|
| <b>Important:</b> | Do not use this parameter if the loading contains enforced displacements or the results may be incorrect. |
|-------------------|---|

- 0      The job will be aborted before Marc is spawned with a “Severe Warning” message. (Default)
- 1      The job will run to completion (if there are no other errors) and a standard Warning message will be issued.

It is recommended that if the same loads are to be used in multiple subcases that each subcase have a different LOAD ID. A typical file setup for SOL 600 should be setup in the following manner:

```
SOL 600,NLSTATIC PATH=1 STOP=1
CEND
DISP=ALL
STRESS=ALL
SPC=123
TEMP(INIT)=33
PARAM,MARCSAME,1
SUBCASE 1
LOAD=100
```

```
SUBCASE 2
LOAD=200
TEMP (LOAD)=300
BEGIN BULK
LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 1., 1.0, 1000, 1.0, 2000
PLOAD4, 1000, 10, 20.0
PLOAD4, 2000, 20, 25.0
(Other Bulk Data entries)
ENDDATA
```

**Note:**

In the above example if the Bulk Data LOAD entries are changed to:

```
LOAD,100,1.,1.0,1000,1.0,2000
LOAD, 200,0.9,1.0,1000,1.0,2000
```

The loads are the same before the overall scale factor (1.0 and 0.9, respectively) are applied and PARAM,MARCSAME,1 is needed. However if the entries were as follows, it is not needed because the loads before the overall scale factor is applied are different.

```
LOAD,100,1.,1.0,1000,1.0,2000
LOAD,200,1.0,0.9,1000,1.0,2000
```

**MARCSCLR**

SOL 600 only.

Sets property values for stiff beams when PARAM,MARCRIGD or PARAM,MSTFBEM are used in SOL 600.

The value of this parameter scales the “default” properties of stiff beams or plates if parameter MARCRIGD=1. The “default” (unscaled) values for the stiff beams are A = 10, I = 100 (both directions) J = 200, shear area = 5 and plate thickness of 1.5. Linear scaling is used for all areas and thickness, all inertia terms are multiplied by the square of the Value entered.

**MARCSETS**

Integer, Default = 0, SOL 600 only.

Controls the type of sets used for the Marc input file element and node sets.

- |   |   |
|---|---|
| 0 | Sets will have the format DEFINE, ELEMENT, ... or DEFINE, NODE, ... |
| 1 | Sets will have the format DEFINE, ELSQ, ... or DEFINE, NDSQ, ...    |

**Remarks:**

1. PARAM,MARCSETS,1 can speed up certain large Marc jobs by a considerable amount.
2. The parameter may be set in RC files.

## MARCSETT

Default = 0, SOL 600 only.

- 0      The current environment is not printed. (Default)
- 1      The current environment is printed in the .f06 file. A user program named eodenv.f must be compiled, linked and placed in the input file directory. The contents of eodevn.f resembles the following:

```
Program eodenv
call system("set")
stop
end
```

## MARCSINC

Default = 0, SOL 600 only.

This parameter controls how often a spline file is written if the spline option (analytical contact for deformable bodies) is requested. If this parameter is not entered or if it is 0 or -1, then a file is not written. If N is greater or equal to 1, then every nth time step is written. Spline files have the extension \*.mfd which may be processed by MSC Mentat.

## MARCSIZ3, Value

Default = Program determines value, SOL 600 only.

- Value      If entered, the integer value entered here is the third value on Marc's SIZING entry representing the maximum number of elements.

## MARCSIZ4, Value

Default = Program determines value, SOL 600 only.

- Value      If entered, the integer value entered here is the fourth value on Marc's SIZING entry representing the maximum number of grid points.

## MARCSIZ5, Value

Default = Program determines value, SOL 600 only.

- Value      If entered, the integer value entered here is the fifth value on Marc's SIZING entry representing the maximum number of constrained degrees-of-freedom.

## MARCSIZ6, Value

Default = Program determines value, SOL 600 only.

**Value** If entered, the integer value entered here is the sixth value on Marc's SIZING entry representing the maximum number of elements in the largest list of distributed loads (the internal Marc to Nastran translator generates these one at a time, so this value is normally 1).

## MARCSLHT

Default = 5, SOL 600 only.

Number of layers through the shell thickness used to integrate shell and beam elements. For linear behavior, N=1 is sufficient. For most plasticity problems, N=5 is adequate. For extremely nonlinear plasticity problems N=11 should be used. SOL 600 requires that N be 5 or larger. If N is entered with a positive value less than 5, SOL 600 will set it to 5. To use values smaller than 5, enter N as a negative number. The absolute value will be used, however the job may fail or results may be incorrect if the model has plasticity.

|              |   |
|--------------|---|
| <b>Note:</b> | Use of PARAM,MARCDEF can effect the value of Marc's SHELL SECT parameter if PARAM,MARCSLHT is not entered. To eliminate SHELL SECT from the Marc file set N to -9999. |
|--------------|---|

## MARCSOLV

Default = 8, SOL 600 only.

Determines the solver to use for Marc.

- |    |  |
|----|--|
| 0  | The profile solver will be used (solver 8 should normally be used instead) |
| 2  | The Sparse Iterative Solver will be used.                                  |
| 4  | The Direct Sparse Solver will be used.                                     |
| 6  | A hardware-provided solver will be used.                                   |
| 8  | A sparse solver similar to the one used by Nastran will be used (Default)  |
| 9  | The CASI element based iterative solver will be used.                      |
| 10 | The mixed direct/iterative solver will be used.                            |
| 11 | Pardiso direct solver (see param,mrthread)                                 |
| 12 | MUMPS parallel direct solver (see param,mrthread)                          |

|              |   |
|--------------|---|
| <b>Note:</b> | If any NLSTRAT entries are entered, the solver type must be specified using the IOLSVER option of NLSTRAT rather than this parameter. |
|--------------|---|

## MARCSPCC

Default = 0 if no SPCD's are entered or 2 if there are SPCD's in the input.

Determines how SOL 600 will proceed if non-zero value enforced displacements are found in SPC entries for multiple subcase models.

- |   |   |
|---|---|
| 0 | A fatal error will be issued and the user will be requested to change all non-zero value enforced displacements to SPCD's.  |
| 1 | The job will continue without any changes (wrong results could be produced).  |
| 2 | All displacements on SPC entries will be set to blank and the job will continue (will usually produce the correct results if SPCD's with the correct values are specified). |

### Remark:

Non-zero enforced displacements should always be specified using SPCD (not SPC) for SOL 600. It is not necessary to specify SPC's for degrees of freedom where SPCD's are applied for SOL 600, but it will not do harm unless non-zero enforced displacements are specified on the SPC entries (either in addition to being specified on SPCD's or without having any SPCD's). If there is only one subcases, the non-zero enforced displacements may be specified on SPCD's and/or SPC's, however for multiple subcases wrong results can result even if the displacements specified on the SPC's and SPCD's are the same. The results will be even further off if the displacements are different. The remedy is to ensure that the SPCD's reflect the correct applied displacements, and then either remove the applied displacements from all SPC's or to set MARCSPCC=2.

## MARCSTIFF, Time

Default = 1.0, SOL 600 only.

- |      |  |
|------|--|
| Time | This parameter specifies what time matrices entered using PARAM,MARCFILi will be used in a Nastran solution. The file may contain matrices at several times, but only the matrices specified by the parameter will be used. This parameter is not usually used, MRMTXNAM,NAME is used instead. |
|------|--|

## MARCSTOP

SOL 600 only.

- |   |   |
|---|---|
| 0 | Normal Marc execution when spawned from Nastran.                                      |
| 1 | Marc will exit (with exit code 7) after phase 0 (corresponds to Marc parameter stop). |

## MARCSUMY

Default = 0, SOL 600 only.

Determines if the summary of maximum values is to be printed.

- 0 A summary of maximum displacements, stresses and strains will be printed in the Marc output file. (Default)
- 1 The summary of maximum values is not output.

## MARCT16

Default = 2, SOL 600 only.

Controls generation of a Marc t16 file. MARCT16=0, Marc does not generate a .t16 output file.

- 1 or 0 Does not generate a t16 file.  
Parameter omitted generates a t16 file. All entries are controlled by the MARCOUT Bulk Data entry, or if MARCOUT is not specified, by the default shown in option 2 below.
- 1 Generates a .t16 output file with the following post codes:  
 11 11,1 11,N 12 12,1 12,N 13 13,1 13,N 14 14,1 14,N  
 15, 15,1 15,N 16 16,1 16,N 17 17,1 17,N 18 18,1 18,N  
 7 7,1 7,N 27 27,1 27,N 301 301,1 301,N 321 321,1 321,N  
 341 341,1 341,N 401 401,1 401,N  
 Nodal: 1, 2, 3, 4, 5, 6, 35, 36, 37, 38, 39, 40, 46, 48, 51, 52
- 2 Generates a .t16 output file with the following post codes. (Default)  
 301, 301,1 301,N 341 341,1 341,N 47  
 Nodal: 1, 2, 3, 4, 5, 6, 34, 35, 36, 37, 38, 39, 46, 48
- 3 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):  
 301, 341 47  
 Nodal: 1, 2, 5, 6, 34, 35, 36, ,37, 38, 39
- 4 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):  
 301 341 47  
 Nodal: 1

### Remarks:

1. This entry is used as an easy way to control which results are placed on Marc's op2 file. All entries can be overridden using the MARCOUT Bulk Data entry. For MARCOUT values of 1 and larger, MARCOUT should be omitted for the input file. MARCOUT should not be used if PARAM,MARCT16 is entered.
2. The frequency of output is controlled by the NLPARM or Bulk Data entries (variables INTOUT and NO respectively).
3. Items such as 341,1 and 341,N designate stresses at the bottom and top surfaces (for applicable elements). Items such as 341 designate stress at mid-thickness.

4. When stress 341 is specified for models with composite elements, the value will be changed to 391 to obtain stresses in the fiber (layer) direction.
5. Consult Marc Volume C documentation for the meaning of the above blocks. Option 1 provides most of the structural output anyone might want, option 2 provides total strain, Cauchy stress, displacement and contact information at the top, center and bottom of surfaces. Option 3 provides the information of option 2 but only at the center (not at the top and bottom).
6. It is necessary to generate a t16 file in order to produce op2, xdb, f06 or punch results.
7. Op2, xdb, punch and f06 results can only be created using option 1 and 2 although option 0 can also be used if the selected outputs are the same as option 1 or 2.
8. If a Bulk Data VCCT entry is included in the model, nodal post codes 57-64 and 74 will be added to any of the previous defaults if MARCOUT is not entered in the model.

## MARCT19

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | Marc does not generate a .t19 output file. (Default) |
| 1 | Generates a .t19 output file.                        |

## MARCTABL

Default = 0, SOL 600 only.

Determines if contact table is to be generated.

- |   |  |
|---|--|
| 0 | Contact tables will be generated for the main Marc input (phase 0) and for each subcase if specified by the user (Default). When MARCTABL=0, each subcase may have a BC CONTACT Case Control command and a matching BCTABLE ID entry. In addition, Marc's "Phase 0" entry is supported by entering a BCTABLE with an ID of zero (or 1,000,000). Each separate BCTABLE will reference the BCBODY entries defined which, in turn, reference BSURF entries. (Default)                             |
| 1 | No contact tables will be generated and all contact bodies (if any) will be placed in the main input data section. Contact will thus be the same for all subcases. When MARCTABL=1, there must only be one BCTABLE entry in the file. There must only be one BC CONTACT command in the Case Control and it must be above all subcases. The BC CONTACT and BCTABLE entry must have the same ID. The BCTABLE entry can reference several BCBODY entries which, in turn, reference BSURF entries. |

## MARCTEDF

Character\*8, no Default, SOL 600 only.

Enter the Marc nthcnt file name without extension. Use this option only if PARAM,MARCTEDN,1 is entered (the file name without extension is limited to 8 characters). The characters ".nthcnt" will automatically be appended to the name specified.

## MARCTEDN

Default = 0, SOL 600 only.

Determines whether a thermal contact analysis will use the Marc jid.nthcnt file generated by this run or use one input by the user.

- 0 Analysis uses the Marc jid.marc.nthcnt file generated in this run. (Default)
- 1 Analysis uses a Marc nthcnt file generated by the user or in a previous run.

**Note:** If MARCTEDN=1, PARAM,MARCTEDF below must be entered to specify the file name.

## MARCTEMP

Default = 1, SOL 600 only.

- 0 The scratch files produced by Marc will be in the same directory as the Nastran input file.
- 1 The scratch files produced by Marc will be in the same directory as the Nastran scratch files. (Default)

**Note:** The Marc scratch files cannot be split.

## MARCTETT

Integer, Default = 0, SOL 600 only.

Determines how CTETRA elements will be mapped to marc.

- 0 4-node or 10-node Nastran CTETRA elements will be mapped to the Marc tet-4 or tet-10 elements.
- 11 4-node or 10-node Nastran CTETRA elements will be mapped to Marc solid element type 7. for tet-10 elements, the mid-side nodes are neglected. the 4 CTETRA nodes (N1,N2,N3,N4) are mapped to the 8 solid shell nodes as follows N1,N2,N3,N4,N4,N4,N4.
- 12 Same as MARCTETT=11 except debug is turned on (for the tetra conversion only)

### Remark:

MARCTETT should not be greater than zero for heat transfer or Herrmann elements.

## MARCTIEC

Default = 1, SOL 600 only.

- 1 Transient Time Integration Error Check for Marc's AUTO STEP method. A value of 1 turns the check on. (Default)
- 0 A value of 0 turns the check off. Turn the check off to match Marc results for version prior to Marc 2003 r1.

## MARCTOL

Default = 0, SOL 600 only.

Determines the method of convergence tolerance.

If parameter MARCTOL=0, convergence tolerances are based on residuals (loads). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy).

- 1 Convergence tolerances are based on displacement.
- 2 Convergence tolerances are based on strain energy.
- 4 Convergence is achieved when either residual or displacement satisfies the criteria.
- 5 Convergence is achieved when both residual and displacement satisfies the criteria.

## MARCTOTD

Default = 0, SOL 600 only

Determines whether for SOL 600 dynamic analyses will use full table association.

- 0 Full table vs time association is not used. (Default)
- 1 Full table vs time association will be used

### Remarks:

1. param,marctott,1 must also be set if MARCTOTD=1
2. This parameter may be set in RC files.

## MARCTOTL

Default = 0, SOL 600 only

Determines whether total or incremental loads are used in a SOL 600 static nonlinear analysis.

- 0 Incremental loads are used. (Default)
- 1 Total loads are used

**Remarks:**

1. Warning - If MARCTOTL=1, MRFOLLOW3=1 will be set if possible (see PARAM,MRFOLLOW3)
2. This parameter may be set in RC files.

**MARCTOTT**

Default = 0, SOL 600 only

Determines whether total loads, including pressures, gravity, spcd, etc., with associated tables are used in a SOL 600 static or dynamic analysis (Remark 3.)

- |   |   |
|---|---|
| 0 | Total loads with tables are not used. (Default) |
| 1 | Total loads with tables are used                |

**Remarks:**

1. Warning - If MARCTOTT=1, MRFOLLOW3=1 will be set if possible (see PARAM,MRFOLLOW3)
2. This parameter may be set in RC files.
3. For full table association, dynamic analyses require PARAM,MARCTOTD,1 to also be set.

**MARCTUBE**

Default = 0, SOL 600 only.

Determines whether CTUBE maps to Marc element 31 or 98 (Default = 0 if this parameter is omitted).

- |   |   |
|---|---|
| 0 | CTUBE elements map to Marc tube element 31 (see Remark) (Default) |
| 1 | CTUBE elements map to Marc element 98 (elastic beam)              |

**Remarks:**

Marc element 31 does not support thermal loading or creep. If the model contains thermal Loading or creep (even if not applicable to the CTUBE elements, option 1 is always used.

**MARCTVL, Value**

SOL 600 only.

- |       |  |
|-------|--|
| Value | If parameter MARCTVL is entered, it must have a real value. The value entered is the convergence tolerance used by Marc (see PARAM, <a href="#">MARCTOL</a> ). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy). |
|-------|--|

## MARCUSUB, chr

No Default, SOL 600 only.

chr        chr is the name of a “user subroutine” to be included in the Marc run. chr is limited to 8 characters without the .f extension. This file must be located in the same directory as the Nastran input data. The subroutine name must have all lowercase letters. chr should be in lower case to prevent confusion. (Nastran will convert the file name to uppercase, but it will be reconverted to lower case when Marc is spawned.) Any user subroutine available to Marc may be specified. Multiple user subroutines must be combined into one file.

Restriction: The computer must have a Fortran compiler and Linker and the Fortran compiler must be the same as used to create the original Marc executable (see the Marc installation manual). The Marc input file does not as yet call out user subroutines, so manual editing of the Marc input file may be necessary in some cases to invoke them. Existing regular Marc subroutines can be modified and handled in the same manner if available to you.

**Note:** If more than one user subroutine is required, all should be combined into one file before execution.

## MARCVERS

SOL 600 only.

This parameter has been replaced by the following parameters:

param,mrconver  
param,mrcontab  
param,mrsprver

## MARCWDIS

Default = 1, SOL 600 only.

Determines whether Marc parameter section DIST LOADS Is written or not. If any of the three values for DIST LOADS are entered (see PARAM,MARCDIS2 PARAM,MARCDIS3 PARAM,MARCDIS4) it will be written. If PARAM,MARCWDIS,1 is entered, it will be written. If PARAM,MARCWDIS,-1 is entered, it will not be written.

**Caution:** Nastran cannot estimate these values very well and produces overly conservative numbers that sometimes leads to failure of the Marc run due to lack of memory. We suggest that the user should use this parameter sparingly and enter the MARCDIS2, MARCDIS3 and MARCDIS4 values for best Marc memory usage.

## MARCWELD

Default = 0 if this parameter is omitted, SOL 600 only.

Determines how CWELD/PWELD elements will be translated to Marc.

- 0 Add extra nodes and elements to simulate the weld.
- 1 Use Marc CWELD/PWELD formulation (available starting with MD Nastran R2 and MSC Nastran 2007).

This parameter may be set in RC files.

## MARELSTO

Default = 0, SOL 600 only.

Determines whether Marc's parameter ELSTO will be created.

- 1 ELSTO will not be created.
- 0 ELSTO will be created only for large models as determined by Nastran. The value of ELSTO will be 40960 except for parallel runs (where a PARAMARC entry exists in the input file) in which case ELSTO will not be created. (Default)
- >0 ELSTO will be created with the value specified used as the Marc ELSTO parameter whether or not the run uses parallel processing.

## MARGPFEL

Default = 0, SOL 600 only.

Used to determine if Marc GRID FORCE output will occur by element, by node, or both ways.

- 1 Output is by element only.
- 0 Output is by node only. (Default)
- 1 Output is by both element and node.

### Notes:

1. This parameter effects the contents of the Marc jid.marc.grd file. Option 0 or 1 is required for GPFORCE output in op2, xdb, punch, and/or f06 files.
2. This parameter can be set in RC files.

## MARFACEA

Default = 1, SOL 600 only.

Face number for "A" side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

## MARFACEB

Default = 1, SOL 600 only.

Face number for “B” side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

## MARFATAL

Default = 1, SOL 600 only.

Determines whether non-existent grid id's for BCBODY entries will cause fatal errors or not.

- |   |   |
|---|---|
| 0 | Non-existent grid ID in BCBODY entries will only cause warning messages. The grid ID will be reset to zero and the job will continue. The results should be checked carefully. In some cases this could result in an entire contact body being skipped. |
| 1 | Non-existent grid ID in BCBODY entries will cause FATAL ERRORS. (Default)   |

## MARGPFOR

Default = 0, SOL 600 only.

Used to determine whether GPFORCE is active for SOL 600. A Case Control request GPFORCE must be specified to obtain grid point forces (see note 4). The grid point forces are output from Marc on a file named jid.marc.grd. The t16op2 program reads this file, puts the data on the f11 file along with displacements, stresses, etc. After t16op2 finishes, the f11 file is brought into DBALL, from which DMAP generated on the fly can produce op2, xdb, punch, or f06 output.

- |    |   |
|----|---|
| -1 | GPFORCE is ignored (Default).                                 |
| 0  | GPFORCE is output for the last time in the last subcase only. |
| 1  | GPFORCE is output at all times given in the .sts file.        |
| N  | GPFORCE is output for times 1, 1+N, 1+2N, etc.                |

### Notes:

1. This parameter can be set in RC files.
2. Output is available only in the new Marc jid.marc.gid file.
3. The Case Control GPFORCE request must be above all subcases, or the same within all subcases.
4. The jid.marc.grd file can become very large if option 0 or N is not used.

## MARHEATM

Default = 0, SOL 600 only.

Determines whether a file named heatm.rc is necessary to run the second phase of SOL 600 heat transfer initial contact job.

- 0 heatm.rc is not required. Defaults will be used. The defaults are scr=yes batch=no mem=80mw
- 1 A heatm.rc file will be supplied by the user in the same directory as the original Nastran input file. The heatm.rc can contain any information used by other rc files except that batch=no. If the original Nastran input file is named jid.dat (or jid.bdf) and out=jid is specified, the final output will be in files such as jid.f06, jid.op2, jid.xdb. If out=jid is not specified the final output will be in files such as jid.nast.f06, jid.nast.op2, jid.nast.xdb.

## MARHTPRT

SOL 600 only.

Controls heat transfer output in the Marc .out file.

- 0 Do not print any output except for summary table.
- 1 Print the nodal temperatures.
- 2 Print all possible nodal heat transfer output.

## MARIBOOC

Default = 0, SOL 600 only.

- 0 Incremental backup data will be stored in memory. (Default)
- 1 For large problems, incremental backup data will be stored on disk. This option triggers Marc's parameter IBOOC.

## MARIPROJ

Default = 0, SOL 600 only.

Flag to determine if auxiliary nodes of a CWELD will be projected on the model or not. (If this parameter is not entered)

- 0 Nodes are not projected on to the model.
- 1 Nodes are projected on to the model.

## MARLDCMB

Default = -1, SOL 600 only.

Determines whether extraneous loads in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 105, 106.

- 1 Extra loads will not be filtered out.
- 2 Extra Bulk Data LOAD entries will be filtered out early in SOL 600 to save computer time.

**Note:**

If PARAM,MARLDRMV,-1 is not entered, extra FORCE, MOMENT and PLOAD4 entries not used in the present analysis will also be filtered out early in SOL 600. Computer time saving can be appreciable for certain models with many extra loads.

## MARLDRMV

Default = 1, SOL 600 only.

Determines whether extraneous FORCE, MOMENT, and/or PLOAD4 entries in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 103, 105, 106.

- 1      Extra loads will not be filtered out.
- 1      Extra Bulk Data FORCE, MOMENT, and/or PLOAD4 entries will be filtered out early in SOL 600 to save computer time. (Default)

**Note:**

This option will be ignored (same as PARAM,MARLDRMV,-1) if PARAM,MARLDCMB,-1 is entered.

## MARMPCHK

Default = -1, SOL 600 only.

Determines whether Marc parameter MPC-CHECK is written.

- 1      MPC-CHECK is not written. (Default)
- 1      Apply the MPCs in the default order:
  1. MPCs obtained from SERVO LINK option.
  2. MPCs obtained from INSET option.
  3. MPCs obtained from TYING, RBE2, or RBE3 options (the actual order follows from the order of these options in the model definition block of the data file.)
  4. MPCs obtained from CYCLIC SYMMETRY option.
  5. MPCs obtained from CONTACT option.

Print a warning message if a tied degree of freedom is being used by a subsequent MPC.
- 2      Same as 1, but instead of warning, a fatal error message is printed and the analysis will stop with exit 2001.

- 3 Try to rearrange the MPCs in such a way that a tied degree of freedom will not be used in a subsequent MPC. If this reordering cannot successfully be completed, print a fatal error message and stop the analysis with exit 2011.
- 999 Skip MPC-CHECK entirely, prevent Marc's error message "Exceed the max trial to order RBE2/3". This option should be used in conjunction with PARAM,AUTOMSET,YES only. (Marc feature,3901 is set)

**Note:** This parameter can be set in RC files.

## MARMPCID

Integer, Default = 1, SOL 600 only

Determines the ID of the MPC Case Control command for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see SOL 600 Executive Control statement, CONTINUE option).

- 0 or 1 A line MPC=1 will be inserted in the Case Control (remark)
- N A line MPC=N will be inserted in the Case Control (N must be between 2 and 9999)

### Remark:

When models with contact and if BCPARA,INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc\_xxxx where xxxx ranges from 0001 to the number of increments in the solution. The ID of these MPC's is usually equal to one. If for some reason it is not one, the actual value may be specified by this parameter.

## MARMTLCK

Default = 0, SOL 600 only.

Determines whether a check of various property-material combinations for SOL 600 will be made or not.

- 0 The checks will not be made. (Default)
- 1 Check will be made. These take extra computer processing time and for most models are not required. The user should turn on these check if he is in doubt if any property-type of material combinations entered into the model may be in error. Current check made are for the following illegal combination.  
PSOLID/MATHE (model should be PLSOLID/MATHE)  
PSOLID/MATHP (model should be PLSOLID/MATHP)

## MARNOCID

Default = 0, SOL 600 only.

SOL 600 does not support MCID defined by cylindrical or spherical coordinate systems. This parameter determines whether MCID defined by cylindrical or spherical coordinate systems will be ignored or “fatalized out” for shell and solid elements.

- 0      The run will be “fatalized out” if MCID defined by cylindrical or spherical coordinate systems are found for shell or solid elements. (Default)
- 1      MCID defined by cylindrical or spherical coordinate systems for shell or solid elements will be ignored.

## MARNOSET, Name

No Default, SOL 600 only.

- Name      Character. If entered, this parameter will not write out a set with the specified name. This is useful, when portion of the model specify sets that are not actually used in the present analysis. Up to 20 of these can be specified.

## MAROFSET

Default = 1, SOL 600 only.

Determines how beam and shell offsets are applied.

- 0      Extra grids and rigid elements will be created to model the offsets.
- 1      Marc will automatically handle offsets for beam and shell elements. No extra grids or elements will be created. The offsets will be found in Marc’s GEOMETRY data. (Default)
- 2      Marc will automatically handle offsets for beam elements only.
- 3      Marc will automatically handle offsets for shell elements only.

**Note:** If MAROFSET is 1 or 2, the beam orientation can be specified using the CBAR/CBEAM “BIT” flag. It is suggested that only combinations, GGG or BGG be used.

## MARPLANE

Default = 0, SOL 600 only.

For composite structures described using PCOMP, together with CQUAD4, CQUADR, etc., it is not possible to tell whether a standard 3D shell or a plane strain shell has been modeled. If MARPLANE is set to 1, such composite models will be assumed to be plane strain (as if a PLPLANE property had been entered rather than PCOMP).

## MARPROCS

Determines the number of domains to be used in a DDM parallel execution.

- 1        Use 1 processor.
- 2        Use 2 processors.
- N        Use N processors.

### Remarks:

1. This parameter is not intended to be used by parallel jobs using several different machines.
2. This parameter and PARAMARC may not both be entered in the same run.
3. This parameter and PARAM,MRTHREAD may both be used in the same job.
4. The parameter may be used with most of the solver types except for the MUMPS solver (type 12).

## MARRBAR2

Default = 1, SOL 600 only.

If MARCRBAR is set to 1, all CMA fields will be changed to 123456.

- 0        CMA field will remain as coded by user, Marc run may fail.
- 1        CMA field will be set to 123456. (Default)

## MARROUTT

Default = -1, SOL 600 only.

Determines whether an inconsistent set of outputs between the Marc t16 file (selected using MARCOUT) and standard Nastran output selected using Case Control requests (and param,post) is allowed or not.

- 1        Inconsistent output is not allowed, if the outputs are inconsistent, the job will fatal early before spawning Marc. (Default)
- 1        Inconsistent output is allowed. However, if the output requests are inconsistent, the job may fail during the t16op2 conversion after Marc has finished.

### Remark:

Nastran outputs in the op2, xdb, punch or f06 files are obtained by converting the Marc output from the t16 file. If output is not available in the t16 file, or if it is in the wrong form, errors will occur during the t16op2 conversion. For most problems, if op2, xdb, f06 and/or punch output is required, it is best not to enter MARCOUT. In most cases MARCOUT should only be used if post-processing using the Marc t16 file is to be done.

## MARSHRII

Integer  $\geq 0$ , Default = 0, SOL 600 only.

Controls how the properties of bars added to the edges of CSHEAR elements when f1 or f2 of PSHEAR is nonzero.

- 0 CBAR elements at the edges of CSHEAR elements (if any) will only have area (I1, I2, J are all 0.0). See PSHEAR Remark 2 as to how the area of these bars is calculated.
- > 0 The value specified will be used to calculate I1, I2 and J from the area as  
 $I1=I2=A/MARSHRII$ .  $J = 2.0 * I1$

### **Remark:**

This parameter must be entered in the Case Control above any subcases.

## MARTET10

Integer, Default = 10, SOL 600 only

Controls how to treat badly shaped 10-node tetra elements in SOL 600 if renumbering the element does not correct problems.

- 1 Stop the analysis if all possible re-numbering options do not correct the bad element. See Remark 4.
- 0 Continue the analysis with the best re-numbering option and let Marc possibly fail with inside out messages.
- 1 If all possible re-numbering options fail, change the bad tet 10 element to tet 4. See Remarks 1 and 5.
- 2 If all possible renumbering options fail, deactivate the bad tet 10 element. (See Remarks 2, 3, and 5.)
- 3 Change the coordinates of bad tet 10 midside nodes to lie at the midpoints of the vertices. (See Remark 6.)

### **Remarks:**

1. Changing bad tet 10 elements to tet 4 may ignore continuity on the edges of adjacent tet 10 elements unless they also get changed to tet 4 elements. If many tet 10 elements are changed to tet 4 elements, the accuracy of the stresses could be inaccurate. In addition, if the elements are used in contact, the same contact body can't have mixed elements with midside and without midside nodes.
2. Deactivating bad tet 10 elements creates holes in the model. If only a few bad elements are deactivated in a fine-grid model, the overall behavior will usually be nearly the same as if the elements were good and remained in the model, however the stresses near the deactivated elements may be too large. If many bad elements are deactivated, all analysis results will probably be inaccurate.
3. Setting PARAM,MARTET10,2 will deactivate all types of elements for which Marc gets inside out elements (not just tet 10 elements). This option maps to Marc's 10-DEACT parameter.

4. This parameter is available starting with MD Nastran 2010. Prior to this release, only option 0 was available.
5. Use options 1 or 2 with extreme care and only when all other alternatives including remeshing using the gui have been exhausted.
6. If martet10=3, any midside nodes whose coordinates are changed and lie on the boundary of the structure will redefine the exterior of the structure by a small amount. Also, if the element is used in contact and the stress-free option is invoked, the coordinates may be changed again and the elements may become badly shaped again.

## MARTETIN

Integer, Default = 0, SOL 600 only.

Controls whether additional information messages are output to the .f06 file or not when param,martet10 is set to a positive value.

- |   |   |
|---|---|
| 0 | Do not output additional tet10 information messages.  |
| 1 | Output additional tot10 information messages regarding checking of all tet10 elements. (The .f06 file could be very large using this option.) |

## MARUPDAT

Default = 1, SOL 600 only.

If this parameter is omitted, the Updated Lagrange method will be used if plasticity is involved. Ensure that all elements in the analysis are capable of using the Updated Lagrange method. If not, enter PARAM,MARUPDAT,-1. The Updated Lagrange Method is more accurate for many problems and also runs faster for some problems.

- |    |   |
|----|---|
| -1 | The Total Lagrange solution procedure will be used when Marc is executed from Nastran.  |
| 1  | The Updated Lagrange solution procedure will be used when Marc is executed from Nastran. This corresponds to Marc parameter update. (Default)                   |
| 2  | The updated Lagrange solution with large rotations for beam elements. This corresponds to Marc parameter (see Marc VOL C documentation for details): update,0,2 |
| 3  | The following Marc parameter will be set, in which LARGE DISP need not be specified elsewhere (see Marc VOL C documentation for details): update,0,2,1          |
| 4  | The following Marc VOL C parameter will be set, in which case LARGE DISP need not be specified elsewhere. (see Marc documentation for details): update,0,,1     |
| 5  | The following Marc parameter will be set (see Marc VOL C documentation for details): update,0,1   |

## MARVFCUT

Default = 0.0001, SOL 600 only.

Controls the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored. (Used in SOL 600 radiation heat transfer only.)

## MAUTOSPC

Default = -1, SOL 600 only.

Determines whether Marc AUTOSPC will be added.

|      |  |
|------|--|
| -1   | Do not add AUTOSPC. (Default)  |
| 1    | Add AUTOSPC to Marc's parameter's section. It will remain throughout the run.      |
| 2000 | Turn on AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option   |
| 2001 | Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1       |
| 2002 | Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2       |
| ...  |  |
| 2999 | Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999     |
| 3000 | Turn off AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option. |
| 3001 | Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1.     |
| 3002 | Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2.     |
| ...  |  |
| 3999 | Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999.   |

## MAXAPL

Default = 1000

MAXAPL specifies the maximum number of applied loads to be used as a basis for residual vector calculations. If the number of applied loads exceeds MAXAPL, User Warning Message 9166 is issued, and residual vector augmentation is disabled. Note that the computing requirements for a large number of residual vectors can be prohibitive. See also, the RESVEC Case Control command.

## MAXDAMP

Default = 1000

See [BEIGRED](#).

## MAXIREVV

Default = 1, SOL 600 only.

Determines whether to reverse axisymmetric coordinates ( $x \rightarrow y$   $y \rightarrow x$ ) and reorder nodes in the elements or not.

- 1      Do not reverse coordinates and renumber elements
- 1      Reverse coordinates and renumber grid ID's for each element. (Default)

### Remark:

Experience has demonstrated that certain models require the -1 option and others require the +1 option. The majority require the +1 option. If one of the options produces inside out messages in the first increment or other error messages, switch to the other option.

## MAXLP

Default = 5

MAXLP specifies the maximum number of iterations for element relaxation and material point subincrement processes in SOLs 129 and 159. MAXLP is 10 in SOLs 106 and 153 and cannot be changed by the user.

## MAXRATIO

Default = 1.E7

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (non-positive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAYOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAYOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

The value -1 of BAYOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101 through 200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence. (See [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.)

BAYOUT and MAXRATIO may be specified in the Case Control Section in order to provide different values between superelements but not between different boundary conditions.

## MBENDCAP

Default = 1, SOL 600 only.

Determines how PBEND internal pressure will be treated.

- 1 Stress stiffening and axial loading due to pressure is ignored
- 0 Stress stiffening and axial loading occurs (this usually happens with end caps)
- 1 Only stress stiffening occurs. (Default)
- 2 Stress stiffening and axial loading occurs for CBEND elements with any nodes having SPC or SPC1 and stress stiffening occurs for all other CBEND elements regardless whether the SPC/SPC1 is referenced above the subcase level or within a subcase.

## MCNLPARM

Default = 1, SOL 600 only.

Determines whether to make all field 5 (KMETHOD) consistent if more than one NLPARM entry occurs in the input stream. If mixed KMETHOD fields are provided for different NLPARM entries, SOL 600 can sometimes encounter difficulties.

- 1 KMETHOD will not be changed to be consistent.
- 1 KMETHOD will be changed to be consistent with that used with the NLPARM entry with the lowest ID. (Default)

### Remark:

This parameter may be set in RC files.

## MCON2D3D

Integer, Default = -1, SOL 600 only.

Controls whether a mixture of 2D and 3D contact will be allowed in a SOL 600 model. For most models a mixture of 2D and 3D contact will fail but there are a few cases where it will run properly. The default will stop the run should a mixture of 2D and 3D contact as specified in field 3 of the first BCBODY line is found. PARAM,MCONF2D3D,1 will allow the job to continue but the job may fail later in the run.

- 0 Stop job with a FATAL ERRO if a mixture of 2D and 3D contact is found.
- 1 Allow job to continue if a mixture of 2D and 3D contact is found.

## MCORDUPD

Default = 0, SOL 600 only.

Determines if the coordinates will be updated if one of the CONTINUE options is specified on the SOL 600 Executive Control statement.

- 0 Original coordinates are used for the continue option. (Default)
- 1 Coordinates are revised to reflect the final position after the nonlinear analysis. These updated coordinates are used in the continue option.

### Remark:

If mcordupd=1, parampmarct19,1 is automatically set and jid.marc.t19 will be generated (jid.marc.t16 may also be generated if desired).

## MCSSHLC

Default = 1, SOL 600 only.

Case Control parameter that determines whether to check that the thickness direction of CSSHL elements is in the direction pointing from grids 1-2-3-4 to 5-6-7-8 or not.

- 1      Do not make the check.
- 1      Make the check. (Default)
- 2      Make the check only for 6-sided elements (looking like chexa) and ignore for 5-sided elements (looking like cpenta).

#### **Remark:**

This parameter must be set in the Case Control above the first subcase.

#### **MCSSHORR**

Default = 1, SOL 600 only.

Case Control parameter that sets the orientation of the thickness direction of CEHXA converted to solid shells.

- 1      The smallest CEHXA direction will be determined and the element will be renumbered such that the thickness of the renumbered elements points from grids 1-2-3-4 to 5-6-7-8
- 1      The thickness direction will be in the grid 1 to 5 direction (thickness points from grids 1-2-3-4 to 5-6-7-8). (Default)
- 2      The thickness direction will be in the grid 1 to 2 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,5,6,2,4,8,7,3)
- 3      The thickness direction will be in the grid 1 to 4 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,4,8,5,2,3,7,6)
- 11     The thickness direction will be along the side closest to the basic X-direction
- 12     The thickness direction will be along the side closest to the basic Y direction
- 13     The thickness direction will be along the side closest to the basic Z direction.

#### **Remarks:**

1. This parameter must be set in the Case Control above the first subcase.
2. This entry is only used in conjunction with the CSSHLM entry only.
3. This entry should only be used if all elements converted to solid shells are CHEXA, none should be CPENTA.
4. See the CSSHL entry for a description of grids 1 to 8.
5. For options other than mcsshorr=1 the material properties and material orientation direction must be orientated properly.

## MDAREAMD

Default = 1, SOL 600 only.

Option to modify or not modify all DAREA entries which are not associated with any other loads (DAREA entries that supply the actual load).

- 1      Do not modify any DAREA entries.
- 1      Modify all DAREA entries that are not associated with any other load entries and supply the actual loading. (Default)

## MDEFSEPP

Integer, Default = see below

This SOL 600 parameter allows an easy way to set the defaults for contact separation control. Since this parameter may be placed in RC files, the user can set default separation controls appropriate to the type of problems normally run and apply the same values to all models without having to enter them in the bulk data for each run. These values are normally entered on the BCPARA entry. The form of MDEFSEPP is as follows:

PARAM,MDEFSEPP,AABBBBCD

where AABBBBCD is a packed integer field comprised of the following 4 controls:

|      | Variable | Standard Default | Values that can be Entered | Max Digits | Min Digits |
|------|----------|------------------|----------------------------|------------|------------|
| AA   | NODSEP   | 5                | 0 to 99                    | 2          | 0          |
| BBBB | MAXSEP   | 9999             | 0 to 9999                  | 4          | 4          |
| C    | ICSEP    | 0                | 0 to 3                     | 1          | 1          |
| D    | IBSEP    | 0                | 0 to 4                     | 1          | 1          |

### Remarks:

1. All values of AA, BBBB, C, D override the standard defaults.
2. NODSEP, MAXSEP, ICSEP, IBSEP entered on BCPARA override values on this parameter.
3. This parameter can be set in RC files.
4. Max digits is the maximum number of digits that can be entered.  
Min digits is the minimum number of digits that must be entered if this parameter is used.
5. A common setting for moderately difficult separation problems is MDEFSEPP=5001004

AA=NODSEP=5

BBBB=MAXSEP=10

C=ICSEP=0

D=IBSEP=4

6. A common setting for more difficult separation problems is MDEFSEPP=2999934

AA=NODSEP=5  
BBBB=MAXSEP=9999  
C=ICSEP=3  
D=IBSEP=4

## MDK4OPT

Default = 1

Performance parameter used in MDACMS for the calculation of K4. If MDK4OPT>0 the most common value of GE for structure elements will be internally applied to PARAM,G to speed up the calculation of K4. Set MDK4OPT to 0 to disable this feature. The traditional K4 calculation will be used instead.

## MDOPT14

Default = NO

Option 14 of the MDENZO functional module modifies degree of freedom (DOF) based domain decomposition such that DOF belonging to any one grid point are retained in a single domain. This action (PARAM,MDOPT14,YES) lends stability to the resulting solution at the possible expense of some efficiency. Setting MDOPT14 to NO bypasses this operation so that DOF may be distributed to any domain without restriction. This parameter applies to Matrix Domain ACMS.

## MDOTM

Default = AUTO

The default selects the most efficient method for data recovery in DOMAINsolver ACMS (PARTOPT=DOF). There are two methods: (1) Output transformation matrix (OTM) method is selected if the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) exceeds the number of degrees-of-freedom at which displacements are required in order to complete data recovery (DISP, STRESS, etc.) This method also requires PARAM,SPARSEPH,YES which is the default. This method may be forced with PARAM,MDOTM,YES. (2) If the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) does not exceed the number of degrees-of-freedom at which displacements are required in order to complete data recovery condition is not satisfied then the non-OTM method is selected. This method may be forced with PARAM,MDOTM,NO.

## MDOTMFAC

Default = 20

See MDOTM.

## MDREDOPT

Default = -1

This parameter is used in ACMS for matrix reduction operations. It affects the way in which matrices are reduced to modal coordinates. Quantities affected are damping, loads, and acoustic coupling. By default, the ACMS-generated Q-size matrices are reduced to H-size (modal coordinates). This is the most efficient method. If MDREDOPT=1, a triple matrix multiplication is used.

## MDUMLOAD

Default = 0, SOL 600 only.

- 0      For subcases with no applied loads and enforced displacement or velocity of rigid contact bodies, a small magnitude dummy load will be created. Without these dummy loads, Nastran becomes confused and does not produce the correct contact information. If there is only one subcase, the dummy loads are not necessary. (Default)
- 1      Dummy loads will not be produced for any subcase and rigid contact with enforced motion may be incorrectly described.

## MECHFIL

Default = 1.E-6

Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constraint modes being retained. This parameter is used for PARAM,MMMETH,OLD. See also [MECHFIX, 898](#), [MMMETH, 908](#), and [MMFIL, 908](#).

## MECHFIX

Default = AUTO

Control for fixing the massless mechanism problem. This capability is provided automatically for the default of this parameter, listed above. This capability is executed only when the eigensolution does not provide answers because of symptoms consistent with the presence of massless mechanisms. If MECHFIX is set to YES, the constraint modes are removed before attempting an eigensolution. When set to NO, the capability is blocked, the eigensolution will fail, and User Fatal Message 9119 will be printed in the F06 file. See also [MMFIL, 908](#) and [MMMETH, 908](#).

## MECHPRT

Default = NO

For SOL 103 only, if massless mechanisms are found the constraint modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINT MODES, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model and a goal is to remove all massless mechanisms before starting production analysis. The number of each “mode” matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes the output file will be large. There is no method to plot these shapes at present.

## MESH

Default = NO

If MESH=YES is specified, then a summary of the shading status and the subelement mesh for each CHBDYi element referencing a VIEW Bulk Data entry is printed.

## METHCMRS

Default = 0

In dynamic analysis (SOLs 103, 107, 108, 109, 110, 111, 112, 145, 146, and 200), METHCMRS specifies the set identification number of an EIGR or EIGRL entry to be used in the calculation of the normal modes on the v-set of the residual structure.

By default, the residual structure v-set normal modes will be computed based on the METHOD Case Control command selection as long as q-set is present.

## MEXTRNOD

Default = 0, SOL 600 only.

Determines whether extra grids will be added to SOL 600 parallel analyses.

- 0      Extra grids will not be added. (Default)
- 1      Extra grids will be added so that all grids from 1 to the highest grid are defined this was necessary for certain version of Marc prior to the 2005 version. All extra grids that are added have coordinates of 0.0 in all three directions.

## MEXTSEE,N

Default = 0, SOL 600 only, Case Control Parameter

Determines whether SOL 600 external superelement residual runs contain loads in the main input file or not.

- 0      External Superelement residual runs will not contain any loads. (Default)
- 1      External Superelement residual runs will contain all loads.

## MFASTCMP

Default = 1, SOL 600 only.

Determines default composite shell integration method.

- 1      Standard integration method is used. Failure, plasticity, thermal loads, etc. are all allowed. (Default)
- 2      A “fast integration” through the thickness technique is used which ignores thermal strains, plasticity, and temperature-dependent material properties (which should not be entered in the model).
- 3      A “fast integration” through the thickness technique is used which only ignores plasticity. Thermal strains and temperature-dependent materials are allowed.

### Remarks:

1. To override the default integration method, use Bulk Data entry PCOMPF.

2. If option 2 is used and temperature loading is present in the model, the option will automatically be re-set to 3.

## MFEA5701

Default = 1, SOL 600 only.

Determines whether feature 5701 is written to the Marc file. When this feature is set to 1, it disables default rotation checking set to 0.001, which was initially set for early versions of Marc when RBE2 and similar elements were added because the 0.001 value proved to be responsible for convergence problems. For models that need rotation checking, you can enter the value using the NLSTRAT variable RLROTT. We recommend always using MFEA5701,1 to turn off the default 0.001 checking value in Marc whether or not the value is entered using NLSTRAT RLROTT.

- 0      Rigid rotation checking of 0.001 is turned on.
- 1      Rigid rotation checking of 0.001 is turned off. (Default)

## MFORCOR1

Default = 1, SOL 600 only.

Option to correct forces entered twice (at the same node) in multiple subcases. This is not commonly found in the input files.

- 0      Do not correct the forces.
- 1      Correct the forces. (Default)

### Remark:

This parameter can be set in RC files.

## MFORDUPE

Default = 0, SOL 600 only.

Controls how duplicate forces encountered for the same load case are handled in SOL 600.

- 0      Forces will be translated to Marc as encountered even if duplicates are present. In the case of duplicates (more than one set of forces for the same node in the same subcase), all forces found will be translated directly to the Marc input. (Default)
- 1      If duplicates exist, only the last set of forces will be translated.

**Note:** To prevent confusion, it is suggested that duplicate forces not be used.

## MFSKIPPP

Integer, Default = 0, SOL 600 only.

Controls memory near the end of SOL 600 translations will be freed or not.

- 0      Memory is freed.
- 1      Memory is not freed (can save computer time, and, in some cases may be necessary for some models.)

## MGAPINIT

Integer, Default = 0, SOL 600 only.

Controls whether CGAP elements are initially open or closed (during Marc increment zero).

- 0      Gaps are initially open.
- 1      Gaps are initially closed.

## MGLUETOL

Real, Default = 1.0E-6, SOL 600 only

Determines the tolerance for the PERMGLUP option of the SOL 600 Executive Control statement for situation where the primary method (same as SOL 101 with permanent glue) fails to find any glued contact possibilities (for example, for edge contact will usually find no glued contact using SOL 101). If not glued contact is found using the primary method a secondary method, which can sometimes take a significant amount of computer time for large models is used. The second method evaluates all grids that have the same coordinates as the grids within the tolerance specified by this parameter. They will be tied together unless they have SPC's, MPC's or are grids attached to rigid elements.

## MHEATSHL

Default = 0, SOL 600 only.

Determines whether a membrane or thick shell element formulation is used for heat transfer. This parameter can be overridden by individual PSHELL entries. In the current release, the membrane elements should not be used for thermal contact analysis.

- 1      All "quad" elements in the model will have membrane capability regardless of MIDi values on the PSHELL entries.
- 0      All "quad" elements in the model will have membrane capability. (Default)
- 1      All "quad" elements in the model will have thick shell capability (shell sect, 1).
- 2      Shells use 2 dof per node, linear variation of temperature through thickness (shell sect, 2).
- 3      Shells use 3 dof per node, quadratic variation of temperature through thickness (shell sect, 3).
- >3     Shells use  $2*n+1$  dof per node (where ni is the value of MHEATSHL specified). (shell sect,  $2*(n-2)+1$ )

**Notes:**

1. Membrane capability in heat transfer means that the temperature is constant throughout the thickness. The MHEATSHL=0 option can be overridden by entering a non-blank value for MID2, MID3 and/or MID4 on an applicable PSHELL entry in which case the MHEATSHL=1 option will be used if no MHEATSHL parameter is entered.
2. Post options will be added to be the same as the value for Marc's shell sect. For example, if shell sect, 3 is generated, post codes 9,1 9,2 and 9,3 will also be generated.

**MHEATUNT**

Default = 2, SOL 600 only.

Specifies the units for heat transfer using SOL 600.

- 0 SI mm units used
- 1 SI m units used
- 2 US units used. (Default)

**Note:** This parameter is used by Marc's ISOTROPIC (heat transfer) third datablock, fourth field.

**MHEMIPIX**

Default = 500, SOL 600 only.

Controls the number of pixels used in radiation heat transfer for SOL 600 using the hemi-cube method.

**MHOU BOLT**

Default = 0, SOL 600 only.

- 0 SOL 600 transient dynamics will use the single step Houbolt numerical integration method. (Default)
- 1 SOL 600 transient dynamics will use the Newmark Beta numerical integration method.
- 2 SOL 600 transient dynamics will use the standard Houbolt numerical integration method.
- 7 SOL 600 transient dynamics will use the generalized alpha (Hilber-Hughes Taylor) numerical integration method.

**MHRED**

Default = NO

MHRED=YES, selects the c-set and r-set component mode reduction method suggested by Dr. Arya Majed and Ed Henkel. See the *MSC Nastran V2004 Release Guide* for details.

## MICRO

Default = 10, SOL 700 only

Defines the accuracy of the initial conditions in Eulerian elements, when using the geometrical shape definition.

### Format:

PARAM,MICRO,VALUE

### Example:

PARAM,MICRO,15

VALUE            Micro-zoning parameter. (Integer > 0)

### Remarks:

1. MICRO is the number of micro zones into which an element is subdivided during initial condition generation.
2. The default MICRO = 10 results in material fractions as accurate as 0.001. If a higher accuracy is required, a greater value for MICRO can be used, but the CPU time for the generation increases rapidly.
3. Micro zoning is only used when the initial conditions of the Eulerian material are specified on a TICEUL1 entry.

## MINCLDSB

Integer, Default = 0, SOL 600 only

Determines whether the SB value in field 4 of MATF will be included or not in the stiffness formulation of the model (Marc COMPOSITE datablock 3 field 4):

- 0        SB will not be included.  
1        SB will be included.

### Remarks:

1. For releases prior to MD Nastran 2010, SB was not included.
2. If SB is included, Marc parameter TSHEAR will also be included.

## MINIGOA

Default = NO

Allows for the reduction in the amount of disk space used when using superelements. When this parameter is set to YES, the transformation matrix GOA will contain terms only for the degrees-of-freedom in the U5 (USET, USET1, SEUSET, SEUSET1) set. This can allow for a significant reduction in the amount of disk

space used by the database. The limitation of using this approach is that data recovery will be available only for these degrees-of-freedom and elements connected to them.

### **MINRECCC, N**

Default = 0, SOL 600 only.

- N            Integer. Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

### **MINSOUTT**

Default = 0, SOL 600 only

Determines elements that deform so much that they go inside-out in an analysis will be deactivated.

- 0            Issue an error message and terminate the job if an element goes inside-out. (Default)  
1            Deactivate any element that goes inside-out and continue the analysis. This option is particularly useful for models which allow failure (for example those using MATF).

### **Remark:**

param,minsoutt,1 maps to Marc's parameter IO-DEACT.

### **MINVASHF**

Default = 1.0, SOL 600 only

Inverse Power "auto shift" value. A new shift point (in frequency squared) is determined as the highest frequency squared plus this entry times the difference between the highest and next highest distinct frequency squared.

### **MINVCITR**

Default = 40, SOL 600 only

Inverse Power method, number of iterations.

### **MINVCSHF**

Default = 0.0, SOL 600 only

Inverse Power shift frequency in Hz.)

### **MINVCTOL**

Default = 1.0E-5, SOL 600 only

Inverse Power convergence tolerance.

### MINVFMAX

no Default, use MINVNMOD, SOL 600 only

Real. Inverse Power max frequency to extract in Hz.

### MINVNMOD

Default = 5, SOL 600 only

Integer. Inverse Power max number of modes to extract.

### MLDALLOW

Integer, Default = 0

Determines whether forces (FORCE, MOMENT, PLOAD4, etc.) are allowed in SOL 600 if their ID is the same as the value N of any Case Control LOAD=N entry.

- |   |  |
|---|--|
| 0 | Forces with the same ID are not allowed. |
| 1 | Forces with the same ID are allowed.     |

### Remarks:

1. MLDALLOW=0 is the only option available for most other Nastran solution sequences.
2. MLDALLOW=1 was the only option available in SOL 600 prior to MD Nastran 2010.
3. MLDALLOW=1 cannot be used with PARAM,MNASTLDS if MNASTLDS>0.
4. This parameter can be set in RC files.

### MLSTRAIN

Default = -1, SOL 600 only.

Corresponds to Marc's "LARGE STRAIN" parameter and optionally allows large rotation of beams and shells.

- |    |  |
|----|--|
| -1 | Small strain, small rotation analysis is used. (Default)   |
| 1  | Large strain and large rotation analysis of beams and shells is implemented in a way to automatically select the best options for a large strain analysis based upon the element type (see the following table). |
| 2  | Use large strain and large rotation analyses of beams and shells for hyperelasticity and multiplicative plasticity with radial return mapping  |
| 11 | Same as option 1 except beams and shells are limited to small rotations (see Remark 1.)  |
| 12 | Same as option 2 except beams and shells are limited to small rotations (see Remark 1.)  |

| Element Type/<br>Material Model             | 1-Dimensional  | Plane Stress or<br>Membranes or Shell<br>Elements                  | Plane Strain or<br>Axisymmetric, or 3-<br>Dimensional<br>Displacement Form                        | Plane Strain or<br>Axisymmetric, or 3-<br>Dimensional Hermann<br>Form    |
|---|--|--|---|--|
| Conventional<br>elastic-plastic             | Updated Lagrange<br>additive plasticity;<br>no finite strain | Updated Lagrange<br>additive plasticity;<br>includes finite strain | Updated Lagrange<br>additive plasticity;<br>includes finite strain<br>utilized constant<br>strain | Updated Lagrange<br>multiplicative plasticity;<br>includes finite strain |
| Mooney, Ogden,<br>Gent, or Arruda-<br>Boyce | Total Lagrange   | Total Lagrange   | Updated Lagrange  | Updated Lagrange   |
| Foam  | Total Lagrange   | Total Lagrange   | Updated Lagrange  | Updated Lagrange;<br>incompressibility<br>neglected                      |

**Remarks:**

- Starting with MD Nastran 2010, if MLSTRAIN=1 or 2, feature,2103 will also be added to provide the best large rotation solution of beam and shell models using updated Lagrange. To prevent feature,2103 from being used MLSTRAIN can be set to 11 or 12 instead of 1 or 2 respectively if large strain small rotation is desired.
- Starting with MD Nastran 2010, if hyperelastic materials are found in the model, PARAM,MLSTRAIN,12 will be set automatically unless a different value of MLSTRAIN has been set by the user.

**MLSTRAN2**

Integer, Default = 0, SOL 600 only

Determines how the LARGE STRAIN option is written in the Marc input file for SOL 600.

- |   |   |
|---|---|
| 0 | LARGE STRAIN, (other options if applicable) |
| 1 | LARGE STRA, (other options if applicable)   |

**Remark:**

Since SOL 600 writes the Marc input file using large-field free-format input either option will work properly. If the input file is modified by hand and this entry is changed to fixed-format, the LARGE STRA option must be used.

**MMAT2ANI**

Default = 2, SOL 600 only.

Determines how MAT2 will be mapped to Marc.

- |   |   |
|---|---|
| 0 | MAT2 will be mapped to Marc's ORTHOTROPIC option. |
|---|---|

- 1 MAT2 will be mapped to Marc's ANISOTROPIC option.
- 2 MAT2 will be mapped as explained in the following note. (Default)

**Note:** The default, MMAT2ANI=2, maps MAT2 to Marc's ORTHOTROPIC option if G13 and G23 are both zero or blank and to Marc's ANISOTROPIC option if G13 and/or G23 are non-zero.

## MMBOLTUS

Integer, Default = -1, SOL 600 only

Controls how the top and bottom nodes are placed in the Marc "tying 69" input when MBOLTUS is used in a SOL 600 model.

- 1 Same as pre-MD Nastran 2010 versions tying 69 node order is top, bottom, control
- 1 Reversed - tying 69 node order is bottom, top control

### Remarks:

1. If it appears that a bolt preload may be backwards for bolts modeled using MBOLTUS, this parameter may be entered to reverse the mpc equations internally generated in Marc by tying 69 instead of reversing the loads.
2. For the MBOLTUS entry to be consistent with the SOL 400 BOLT entry PARAM,MMBOLTUS,1 is usually required.

## MMEMDETT

Default = -1, SOL 600 only.

Determines the Nastran portion of SOL 600 determines how much usable memory is available just before spawning Marc and informs Marc through the -ml flag. If Marc needs more memory than this value, it will go out of core.

- 1 Do not determine the amount of usable memory. (Default)
- 1 Determine the amount of usable memory and set the -ml Marc command line option to that value

### Remarks:

1. Marc can determine this memory, however the computations determine the amount of installed memory and do not take into account memory used by the operating system or by other user processes.
2. This parameter may be entered in RC files.

## MMFIL

Default = 1.E-10

Filter value used to distinguish between massless mechanism modes and rigid body modes. A smaller value may discard rigid body modes. The default value has been effective on all problems solved to date. This parameter is used for PARAM,MMMETH,NEW and PARAM,MMMETH,BETA. See also [MMMETH, 908](#).

## MMMETH

Default = NEW

Selects method for searching for massless mechanisms. The default value is generally the most reliable method. Use PARAM,MMMETH,BETA to invoke a new procedure that can sometimes find massless mechanisms when the default does not find any. Specify PARAM,MMMETH,OLD to use the original procedure. See also [MECHFIX, 898](#).

## MNASTLDS

Integer, Default = See Remark 1., SOL 600 only.

Option to determine complex forces using OLOADS datablock or the foes, moments, pressures, etc. in the Nastran input file. Positive values of MNASTLDS may also significantly speed up the SOL 600 translation and, in the case of RFORCE, the marc portion of the run will also usually be faster.

- |         |  |
|---------|--|
| -1 or 0 | Loads for Marc are translated or calculated directly from the forces, moments, pressures, etc. in the Nastran input file. (Remark 1.)  |
| 1       | Complex loads and moments are obtained from the OLOADS datablock, pressures, rforce and other distributed loads are translated from the values found in the Nastran input. (Remark 3.) |
| 4       | Same as MNASTLDS=1 if the model has Bulk Data LOAD entries with more than one term (L2, L3, ... on the Bulk Data LOAD entry is defined), otherwise same as mnastlds=0.                 |
| 99      | Same as MNASTLDS=1 regardless of whether complex loads or Bulk Data LOAD entries with more than one term exist in the mode.  |
| 777     | Same as MNASTLDS=99 except that all type of distributed loads (pressures, rforce, etc.) will be converted to force and obtain from the OLOADS datablock. (Remark 5.)                   |

### Remarks:

1. Default=99 if any GRID entry has field 7 defined. Default=777 if there are multiple subcases and RFORCE or PLOAD4 with continuation lines exist in the model or if any FORCE entry with nonzero (or blank) field 4 is defined, otherwise the default is 4.
2. MNASTLDS=-1 (or 0) was the only option prior to MD Nastran 2010.
3. Complex loads are defined as those forces or moments whose grid appears more than once when the Case Control and Bulk Data LOAD entries are evaluated. For example consider the following situation:

```
....  
SUBCASE 1  
LOAD = 10  
SUBCASE 2  
LOAD = 20  
BEGIN BULK  
LOAD, 10, 1.0, 2.0, 1, 3.0,2  
LOAD, 20, 1.0, 4.0, 1, 5.0,2  
FORCE,1,51, ,1.0,0.0,0.0,100.0  
FORCE,2,51, ,1.0,0.0,0.0,200.0
```

Although the force at grid 51 occurs for different load ID's when combined using the LOAD entries it actually occurs twice in each subcase. This is the simplest form of a complex load and can be handled by either MNASTLDS option. However, some models will have much more complex situation, even having different entries for the Fx, Fy and Fz directions. In such circumstances, particularly if multiple subcases are involved MNASTLDS > 0 is recommended.

4. This parameter may be set in RC files.
5. When MNASTLDS=777 pressures (and other forms of distributed loads) are converted to nodal forces and the follower force effects will be lost. However, for multiple subcases the follower force effects are not available in SOL 600 unless the loading is the same between subcases except for a scale factor.

## MODACC

Default = -1

- |          |  |
|----------|--|
| 0        | Selects the mode acceleration method for data recovery in dynamic analysis. See <a href="#">Formulation of Dynamic Equations in SubDMAP GMA</a> in the <i>MSC Nastran Reference Guide</i> for further discussion. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 must also be specified. |
| 1        | Is the same as MODACC = 0 except if SUPPORTi entries are present then the displacements are corrected for rigid body motion and the local flexibility at the SUPPORTi degrees-of-freedom.  |
| $\geq 0$ | Is not recommended for use in hydroelastic problems.   |

## MODEL

Default = 0

This parameter also allows several models to be stored in the same graphics database created by PARAM,POST,0.

## MOFFCORE

Default = 1, SOL 600 only.

Determines how memory for PARAM,MARCOFFT above is to be allocated (for increased speed).

- 1 Additional memory is not allocated.
- 1 Additional memory is allocated if available. (Default)

## MOP2TITL

Default = 1, SOL 600 only.

Determines how titles are placed on the 146 word record for op2 output records generated by SOL 600.

- 1 Standard titles will not be placed, however titles of the form "CQUAD4 STRESS FROM MARC", and similar titles for other element types, strains, displacements, etc. will be written. This option is useful for certain postprocessors.
- 1 Standard Nastran title, subtitles will be placed on op2 files generated by SOL 600. This option is useful for postprocessors that require SOL 600 op2 data to be exactly in the same format as that generated by other Nastran solution sequences. (Default)

**Note:** The SOL 600 op2 file follows that of SOL 109 as closely as possible.

## MOUTRCOR

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 OUTR options are disabled or not if any CORD2i references another coordinate system.

- 0 Disable OUTR options
- 1 Do not disable OUTR options

### Remarks:

1. In some cases, OUTR results are inaccurate if a CORD2i references another CORD.
2. CORD2i entries are not checked to see if they are actually used in the model.

## MP1SET

Character "F" or "G" (default, see below)

Controls the integration set used in the calculation of MONPNT1 results. For the value of "F", the integrated values will reflect any redistribution of loads caused by rigid elements and MPCs. This is particularly useful in the static aeroelastic solution sequences (SOL 144) when RBE3s are used to distribute mass, applied loads and splining. For the value of "G", no redistribution occurs. The default is a function of the solution sequence.

SOL 144 - Default = "F", option "G" is not available.

All other sequences - Default = "G", option "F" can be specified by the user.

## MPCX

Default = 0

See OLDSEQ.

## MPERMPRT

SOL 600 only.

Determines whether to print permanent glue MPC's in the f06 file.

- |   |                                   |
|---|-----------------------------------|
| 0 | Do not print permanent glue MPC's |
| 1 | Print permanent glue MPC's        |

## MPTUNIT

Default=0

MPTUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of monitor points results in OUTPUT2 format in SOL 146. Default=0 indicates that no monitor point data are to be stored

## MQUATERN

Integer, Default = 2, SOL 600 only

Controls whether quaternions will be used for SOL 600 models with large rotation.

- |    |   |
|----|---|
| -1 | Do not use quaternion "feature" for large rotation  |
| 1  | Use quaternion feature,2103 which allows large rotations for total Lagrange or updated Lagrange |
| 2  | Use quaternion feature,2105 which allows large rotations only if updated Lagrange is used       |

### Remarks:

1. The use of MQUATERN = -1 allows backwards compatibility such that most models will obtain the same results using MD Nastran R3 as were obtained using MD Nastran R3.
2. The use of MQUATERN = 1 is believed to yield the most accurate large rotation results for most models.
3. With PARAM,MQUATERN,1 large rotation may be used both with updated Lagrange and total Lagrange.
4. mquatern = -1 does not set Marc feature 2103 (feature 2105 is not set).
5. mquatern = 1 sets Marc feature 2103

## **MRADUNIT**

SOL 600 only.

Controls the units used in radiation heat transfer for SOL 600.

- 1      Degrees Celsius
- 2      Degrees Kelvin (default if parameter not entered)
- 3      Degrees Fahrenheit

**Note:** Degrees Rankin are not available.

## **MRAFFLOR, N**

Default = N0, SOL 600 only.

- N      Integer. If N=0, a new AF\_flowmat file containing temperature-dependent stress-strain curves will be generated during the current Nastran execution and also used in the spawned Marc run. If N=1, an existing AF\_flowmat file will be used. The name of the file is always determined by the value of PARAM,MRAFFLOW, but PARAM,MRAFFLOT determines if other characters are added.

## **MRAFFLOT, N**

Default = N0, SOL 600 only.

- N      Integer. If N=0, the file name as specified using PARAM,MRAFFLOW,Name will be used with no changes except that all characters will be in lower case and the extension “.mat” will be added. If N=1, the characters “asm\_” will be added at the beginning of Name, the first character of Name will be upper case (the other characters of Name will be lower case) and the extension “.mat” will be added. This will make the AF\_flowmat file name compatible with many names in Marc’s AF\_flowmat directory.

## **MRAFFLOW, Name**

No Default, SOL 600 only.

- Name     Character. Name of a file containing temperature dependent stress versus plastic strain curves in Marc’s AF\_flowmat format. This file can be generated from the current Nastran run using TABLEST and TABLES1 entries or a pre-existing file can be used depending on the value of PARAM,MRAFFLOR. The extension “.mat” will be added to Name. If this is a new file, it will be saved in the directory from which the Nastran execution is submitted. If a pre-existing file is to be used, it can either be located in the directory where the Nastran execution is submitted or in the Marc AF\_flowmat directory.

## MRALIAS ID (MALIAS02, MALIAS03, etc.)

No Default, SOL 600 only. This parameter is not usually used.

The purpose of the parameter is to map the advance nonlinear element type selected by the internal Marc translator in Nastran to a different type. For example, if the element type 75 for CQUAD4 is normally used, a mapping to advance nonlinear element type 139 could be made. ID is a 6 digit number. The left 3 digits are the element type normally selected by the translator and the right 3 digits are the element type to be mapped. In the above example, element type 75 is to be mapped to 139. The user would enter ID=075139. If element type 165 is to be mapped to element type 1 (which is not a real case), ID=165001. Consult Marc Volume B for a list of elements and their meaning. The user is responsible for ensuring that the mapping selected is proper. There is a limit of 18 aliases that may be entered in any model. Since Nastran can only accept one parameter with a given name, the second alias should be named PARAM,MALIAS02 and the third PARAM,MALIAS03, etc. All original element types mapped must actually exist in the model.

### Remark:

These entries should only be used if the Marc GEOMETRY entries are identical for the original and new element types.

**Note:** If you use one of the MRALIAS parameters, certain “parameters” in the Marc file may no longer be correct. For example, an element originally capable of using the updated Lagrange method may be aliased to one that must use the total Lagrange method. Such conditions are not checked by the translator when you use alias and you will need to make modifications to the Marc input file yourself to reflect them. To resolve this use PARAM,MLSTRAIN.

The Bulk Data entry, ALIASM, is available and subsequent version are more powerful.

## MRALLOCG

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for general Marc memory when Marc is spawned from Nastran. It specifies the initial allocation of “general memory”. This is used for storing element stiffness matrices and part or all of the matrix solver workspace among other things. Please note that element data like stresses and strains are not part of the general memory. Solvers 6, 8, and 9 use the main part of the workspace in separate memory. Initial allocation of the general memory can be used for avoiding reallocation (increase of the workspace). For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

## MRALLOCS

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for Marc solver memory when Marc is spawned from Nastran. It specifies the initial allocation of memory for solver 8. By giving a value that is more than the maximum used during the run, one avoids that the solver workspace is increased (reallocated). This can be particularly useful for large contact jobs, where additional memory may be allocated due to

contact. If the given workspace is less than what is needed, it is automatically increased. This option is only for use with solver type 8. No check is done to see if solver type 8 is used in the job. For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

### MRBE3SNG

SOL 600 only.

Option to check the singularity of RBE3's in SOL 600. The real value entered is the singularity threshold allowed. If MRBE3SNG is entered as a positive value, all RBE3's with poor singularity values above the value entered will be output in the jid.marc.out file as warning messages. If MRBE3SNG is entered as a negative value, all RBE3's with poor singularity values above the absolute value entered will be output in the jid.marc.out file as error messages and the job will abort if they are found in increment zero, but if they are found after increment zero the messages are warning messages and the job will continue.

**Note:** A "good" RBE3 element using the original geometry in increment zero could become singular as the structure deforms in subsequent increments.

### MRBEAMB

Default = 0, SOL 600 only.

- 0      Write equivalent radius for all beams (see PARAM,BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc's GEOMETRY values for beam type elements. (Default)
- 1     Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.

### MRBEPARM, IJK

Default = 0, SOL 600 only.

IJK provides settings for Marc's RBE parameter. If PARAM,MRBEPARM parameter is entered, Marc's RBE parameter will be set using this IJK. If PARAM<MRBEAMPM is not entered, PARAM,MARCRBE2 or PARAM,MARCRBE3 can be used to set Marc's RBE parameter.

IJK is a combination of three variables. For example 311. Descriptions for individual entries are as follows:

- I      Enter 3 or 6 to control the number of degrees-of-freedom. For the dependent grid (reference grid) of each rbe2 or rbe3. The independent grids can have 3 or 6 dof and can be different than what is specified by I.
- J      Enter 1 to use large displacement formulation of rbe2. Enter 3 to deactivate automatic convergence test for rbe2.
- K      Enter 1 to use large displacement formulation of rbe3. Enter 2 to activate non-normalized rotation constraint coefficient for rbe3.

**Note:** If IJK values other than specified above are entered, IJK will be set to zero and the parameter will not be used. This parameter should not be entered unless there are rbe2's or rbe3's in the model and they are to be used as such in Marc (rather than mpc or stiff beams).

## MRBDYCVT

Default = 0, SOL 600 only.

Determines if CHBDYG is converted to CHBDYE for SOL 600 heat transfer.

- 0      Do not convert. (Default)
- 1      Attempt to convert CHBDYG to CHBDYE. All grids specified by all CHBDYG entries must reference actual conduction elements (CQUAD4, CHEXE, CTETRA, etc.) If even one CHBDYG does not reference an existing conduction element, this parameter will be re-set to zero.

**Note:** If MRBDYCVT=0, CHBDYG will normally result in point heat transfer loads rather than distributed heat transfer loads.

## MRBIGMEM

Integer, Default = 0, SOL 600 only.

- N      If N=0, memory allocations during loads translation phases are sized for computers with limited memory and swap space (paging space). Some large problems and/or unusual problems may not run. If this happens, use a newer modern computer with lots of memory and disk space (and lots of swap space) and set N=1. Larger memory allocations will then be available. This parameter is not usually required unless the available memory is extremely small.

## MRBUKMTH

Default = 2, SOL 600 only

- 1      Buckling modes will be computed using the Inverse Power method.
- 2      Buckling modes will be computed using the Lanczos method. Matrices must be positive-definite for this option. (Default)
- 3      Use Lanczos if EIGRL is specified, Inverse Power if EIGB is specified.

**Note:** MRBUKMTH should be specified if EIGR or EIGRL is used for buckling unless SOL 600,105 is the solution sequence. An alternative is to use ANALYSIS=BUCK. In SOL 600, it is not possible to compute natural frequencies and buckling modes in the same run. If ANALYSIS=BUCK is specified anywhere in the Case Control and if PARAM,MRBUKMTH is omitted, param,mrbukmth=3 will be set automatically.

## MRC2DADD

Default = 0, SOL 600 only.

Allows an offset to be added to all coordinates for 2D analyses so that X and Y will always be positive.

- 0      Offsets will not be added. (Default)
- 1      Offsets will be determined so that all Marc X and Y coordinates are positive (will exceed 0.1).

## MRCKBODY

Integer, Default = 1, SOL 600 only.

Determines whether each BCBODY will be checked to see if the 2nd line is present when more than 2 lines exist. If the 2nd line is missing for BCBODY entries with more than 2 lines, errors can occur.

- 1      BCBODY entries are not checked.
- 1      BCBODY entries are checked. If the 2nd line is not present for BCBODY entries having more than 2 lines, a 2nd line will be added internally.

### Remark:

This parameter can be set in RC files.

## MRCKLOAD

Integer, Default = 0, SOL 600 only.

Controls whether each subcase has LOAD or DLOAD specified. SOL 600 will usually not run correctly unless each subcase has some type of load (dummy or real). If a subcase only has a change in SPC, MPC, or contact to obtain the correct results a dummy load is usually required.

- 0      No check is made to see if each subcase has a Case Control LOAD or DLOAD entry.
- 1      A check is made to see if each subcase has a Case Control LOAD or DLOAD entry and if any subcase does not have such entry, a fatal error will be issued.

### Remarks:

1. This parameter must be entered in the Case Control above any subcases.
2. This parameter may be set in RC files.

## MRCNADD

Integer, Default = 1, SOL 600 only.

Controls whether PARAM,MRCNVER,11 will be added automatically if any BCTABLE with FBSH, BKGL and/or SEGS sub-headers is found. Also controls whether a BCTABLE with ID=0 will be added if one is not present in the input file.

- 1      Do not change value of PARAM,MRCNVER and do not add a BCTABLE with ID=0 if it is missing.
- 1      Change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Also add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.
- 2      Change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found but do not add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.
- 3      Do not change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.

### Remarks:

1. PARAM,MRCNVER,11 is necessary if BCTABLE sub-headers FBSH, BKGL and/or SEGS exist in the model.
2. BCTABLE with ID=0 is sometimes necessary for certain contact problems. It never hurts to have one as it will be ignored for those contact problems where it is not needed.
3. This parameter may be used in MD Nastran 2010 and subsequent versions. The behavior of previous versions was the same as PARAM,MRCNADD,-1.
4. This parameter may be set in RC files.
5. This parameter, if used, must be placed in the Case Control above any subcases.

## MRCNRES

Default = 0, SOL 600 only.

For restart runs, Marc will produce “continuous” results files (t16/t19) which contain the results of the original run(s) as well as the results of the restart run. If Nastran postprocessing is requested to generate op2, xdb, etc. files, they will also contain the results from prior runs as well as the restart run.

MRCNRES=1, output results files will contain the results of the restart run only.

## MRCNTAB

Default = 0, SOL 600 only.

Determines whether CONTACT and CONTACT TABLE for SOL 600 use table-driven form or not.

- 0      Do not use table-driven form. (Default)
- 11     Use table-driven form of CONTACT and CONTACT TABLE

### Remarks:

1. If MRCONTAB=11 then MRCONVER must also be set to 11
2. This parameter should is not necessary for SOL 600 heat transfer analysis and should not be set

### MRCONUSE

Integer, Default = 0, SOL 600 only

Determines which increment for which contact MPC's will be used for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see the SOL 600 Executive Control statement, CONTINUE option).

- 0        The last increment will be used (remark).  
N        Increment N will be used.

### Remark:

When models with contact and if BCPARA<INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc\_xxxx where xxxx ranges from 0001 to the number of increments in the solution. This parameter allows the user to specify which increment will be used in the continuation run spawned by the primary Nastran run after Marc finished. N must be less than or equal to the number of actual increments.

### MRCONVER

Default = 0, SOL 600 only.

Determines version of Marc to use for CONTACT and CONTACT TABLE for structural analysis.

- 0        Uses enhanced Marc version 9 (Marc 2001 with added fields).  
11       Users Marc version 11 (Marc 2005).

### Remarks:

1. This parameter only affects contact and only options 0 and 11 are available. It is not needed unless BCTABLE with FBSH and/or BKGL is entered.
2. This parameter may be set in RC files.
3. If PARAM,MRCONVER,11 is entered it is usually also necessary to enter PARAM,MRSRVER,11.

### MRCORDS

Default = 0, SOL 600 only.

Determines whether Marc COORD SYSTEM will be added if any CORD1i or CORD2i entries are in the model and if CBUSH elements, orientation vectors or other items requiring coordinate system transformation are present in the model or if PARAM,MRPLOAD4,2 is entered which signifies that PLOAD4 with continuation entries with CID, N1, N2, N3 are to be taken into account in the model.

- 0        Add Marc COORD SYSTEM. (Default)  
 1        Do not add MARC COORD SYSTEM

## MRCQUAD4

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CQUAD4 elements in SOL 600.

- 75        Advance nonlinear element type 75 is used. (Default)  
 139      Advance nonlinear element type 139 is used.

**Note:** Element 75 is capable of thick shell behavior and element type 139 is applicable to thin shells.

## MRCTRIA3

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CTRIA3 elements in SOL 600.

- 75        Advance nonlinear element type 75 with a duplicate node is used. (Default)  
 138      Advance nonlinear element type 138 with 3 nodes is used.

**Note:** Element 75 is capable of thick shell behavior and compatible with the default element 75 used for CQUAD4, but since one node is degenerate (repeated) stresses may not be accurate. Element 138 is accurate for thin shell behavior but may not be accurate for thick shell behavior. Currently there is no triangle thick shell in Marc.

## MRCWANGL

Default = 0.0, SOL 600 only.

Angle in degrees over which to rotate the cross-section about the beam axis to obtain its final orientation.

## MRDELTIT

Default = 3, SOL 600 only.

Determines how delta time is set for each “step” of a SOL 600 transient nonlinear analysis.

- 0        DTI=Ti/N  
 1        DTI=Ttot/N  
 2        DTI=min(Ttot/N, Ti/2.0)

- 3 DTi=min(Ttot/N, Ti/10.0) (Default)
- 4 DTi=min(Ttot/N, Ti/100.0)
- 5 DTi=min(Ti/N, Ti/2.0)
- 6 DTi=min(Ti/N, Ti/10.0)
- 7 DTi=min(Ti/N, Ti/100.0)

1. SOL 600 transient dynamics is run by taking each pair of points on the applicable TABLED1 entry for the applied forcing function as a step. For example, if the following TABLED1 is given, the time steps will be as shown below:

TABLED1, 1  
 +, 0.0, 0.0, 1.0E-3, .2 4.0E-3, .3, 6.0e-3, -.3  
 +, 1.0E-2, 0.0

| Step | Final Time | Delta Step Time (Ti) |
|------|------------|----------------------|
| 1    | 1.0E-3     | 1.0E-3               |
| 2    | 4.0E-3     | 3.0E-3               |
| 3    | 6.0E-3     | 2.0E-3               |
| 4    | 1.0E-2     | 4.0E-3               |

If the entry for this model is (N=100, Ttot=1.0E-2)

DTi is the initial delta time (AUTO STEP (2,1) for the particular step, the six selections of MRDELTTT would give for the DTi values shown in the table below. Note that option 3 gives at least 10 points per step and the old option 0 is probably too conservative.

| Step | Final Time | Time Step (Ti) | DTi for MRDELTTT |        |        |        |        |        |        |        |
|------|------------|----------------|------------------|--------|--------|--------|--------|--------|--------|--------|
|      |            |                | 0                | 1      | 2      | 3      | 4      | 5      | 6      | 7      |
| 1    | 1.0E-3     | 1.0E-3         | 1.0E-5           | 1.0E-4 | 1.0E-4 | 1.0E-4 | 1.0E-5 | 1.0E-5 | 1.0E-5 | 1.0E-5 |
| 2    | 4.0E-3     | 3.0E-3         | 3.0E-5           | 1.0E-4 | 1.0E-4 | 1.0E-4 | 3.0E-5 | 3.0E-5 | 3.0E-5 | 3.0E-5 |
| 3    | 6.0E-3     | 2.0E-3         | 2.0E-5           | 1.0E-4 | 1.0E-4 | 1.0E-4 | 2.0E-5 | 2.0E-5 | 2.0E-5 | 2.0E-5 |
| 4    | 1.0E-2     | 4.0E-3         | 4.0E-5           | 1.0E-4 | 1.0E-4 | 1.0E-4 | 4.0E-5 | 4.0E-5 | 4.0E-5 | 4.0E-5 |

2. If Dti is set using NLAUTO variable TINIT, this parameter will be ignored.

## MRDISCMB

Default = 0 without gravity, =1 with gravity, SOL 600 only.

Determines whether distributed loads, such as pressure, are combined in the Marc input file into one list if the magnitude of the distributed loads are the same. This parameter applies to MSC Nastran 2005 only. For previous MSC Nastran versions, MRDISCMB was 1. Option 0 save Marc memory and processing time.

- 0      Distributed loads with the same magnitude are combined.
- 1      All distributed loads are input individually. Must be used if multiple subcases with the same loadings are present in the model.

**Caution:** This parameter should be used with caution. For multiple load cases, MRDISCMB=0 may produce the wrong results particularly if gravity loading is present. MRDISCMB=1 must be used if multiple subcases with the same loadings are present in the model.

## MRDUPMAT

Default = 1 without gravity loading and -1 with gravity loading, SOL 600 only.

Controls whether to create duplicate materials for elements used by both the Marc pshell (PARAM,MRPSHELL,1 or the smear option on the SOL 600, ID statement) or not. If the user knows in advance that no materials are used by shells and any other types of elements, this step can be skipped to save considerable computer time.

- 1      Check for materials used by shells and other types of elements and create duplicate materials if any are used by both.
- 1     Do not check or create duplicate materials.

## MRDYNOLD

Default = 1, SOL 600 only.

Determines whether dynamic loads created by SOL 600 are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases or uses a new calculation method.

- 1     Dynamic loads are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases.
- 1      Dynamic loads are “total loads” and may be more accurate for multiple subcases. (Default)

## MREIGMTH

Default = 1, SOL 600 only.

- 1      Eigenvalue analysis will be done in Marc using the Lanczos method. (Default)
- 0      Eigenvalue analysis will be done in Marc using the inverse power sweep with double eigenvalue extraction.
- 3      Eigenvalue analysis will be done in Marc using the inverse power sweep with single eigenvalue extraction.

## MREL1103

Default = 0, SOL 600 only.

- 0        Maps CQUAD4 to Marc's element type 11 for plane strain problems. (Default)
- 3        Maps CQUAD4 to Marc's element type 3 for plane stress problems.

## MRELRB

Default = 0, SOL 600 only.

- 0        If BCMOVE with the release option is specified, IDRDBODY (see the BCMOVE entry) will refer to the actual IDs of BCBODY entries. (Default)
- 1        The release option of BCMOVE is used, IDRDBODY (see the BCMOVE entry) will be in the order of occurrence of the BCBODY entries in the sorted Nastran Bulk Data file. For example, if there are two BCBODY entries with ID 12 and 22, the MRELRB=1 option means that you should specify IDRDBODY on the BCMOVE entry using values of 1 and 2. If MRELRB=0, the IDRDBODY values should be 12 and 22.

## MRELSPCD

Integer, Default = 0, SOL 600.

Controls whether SPCD for the second or subsequent subcase represents total (absolute) displacements or incremental displacements (relative to the displacement at the end of the previous subcase). SPCD for the first subcase is always total (absolute).

- 0        SPCD represents relative displacements (same as SPCR) for subcases 2 and above unless PARAM,MARCTOTT,1 has been set, in which case the SPCD's represent absolute displacements.
- 1        SPCD represents absolute displacements for subcases 2 and above (relative displacements for subcases 2 and above must be entered using SPCR).

### Remarks:

1. This parameter may be set in RC files.
2. PARAM,MRELSPCD,1 will automatically set PARAM,MARCTOTT,1 which means total loads using table-driven input will be used in the analysis. This is the only way total (absolute) displacements for subcases 2 and above can be used in Marc.
3. The behavior of SPCD prior to the MD Nastran 2010 release was to consider SPCD as relative displacements for subcases 2 and above.

## MRENUEL

Default = -1 if parameter is not entered and MRENUEL is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUEL is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with largest element number less than approximately 20000). (Default)
- 1 All elements will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All elements will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the maximum element number is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUEL on the SOL 600 statement).
3. MRENUEL should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUEL=1 an equivalence list of original and re-numbered element numbers is output on file elenum.txt.

## MRENUGRD

Default = -1 if parameter is not entered and MRENUGRD is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUGRD is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with grid IDs less than approximately 20000). (Default)
- 1 All grid IDs will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All grid IDs will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUGRD on the SOL 600 entry).
3. MRENUGRD should not be set both on the SOL 600 statement and as a parameter.

4. For MRENUMGRD=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

## MRENUMBR

Default = 0, SOL 600 only.

Determines if both grid and element IDs for SOL 600 will be renumbered or not.

- 0      No renumbering will occur (suggested for models with largest grid ID less than approximately 20000). (Default)
- 1      All grid and element IDs will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt
- 2      All grid and element IDs will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 entry (see MRENUMBR on the SOL 600 statement).
3. MRENUMBR should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUMBR=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

## MRESTALL

Default = 1, SOL 600 only.

Controls rotational restraints for solid element-only models.

- 0      SPCs for DOFs 4-6 will be ignored if entered in the Nastran file
- 1      SPCs for DOFs 4-6 will be included if entered in the Nastran file. (Default)

**Note:** This option might produce an input-data error in the Marc run but is sometimes required if RBEs or other special items are included in the model.

## MRESULTS

Default = 0, SOL 600 only.

If MRESULTS is set to 3, postprocessing of a previously-generated Marc t16 file to a results-only op2 file (normally called an f11 file) will be accomplished. OUTR=f11 and STOP=3 should also be set on the SOL 600 command line. This capability is available starting with MSC Nastran 2004.1.0. If MRESULTS=0 the t16 file from the current job will be processed if requested by OUTR options on the SOL 600 statement.

## MREVPLST

Default = -1, SOL 600 only.

Determines whether 2D plain stress triangular element node numbers will be reversed or not.

|         |                                 |
|---------|---------------------------------|
| -1 or 0 | Do not reverse the node numbers |
| 1       | Reverse the node numbers        |

## MRFINITE

No Default, SOL 600 only.

Controls Marc's FINITE parameter. If entered, with an integer value of 1, Marc's FINITE option will be employed. If this option is entered, parameters MRFOLLOW1, MRFOLLOW3 and MARUPDAT should also be entered. Other parameters to be considered are MARCDILT, MARCASUM and LGDISP.

## MRFOLLOW

Default = 0, SOL 600 only.

- 0      FORCE1, FORCE2, MOMENT1, MOMENT2 will act as non-follower forces. (Default)
- 1      Follower forces entered using FORCE1, FORCE2, MOMENT1, MOMENT2 will be mapped to Marc's new follower force option.
- 1     Follower forces will be turned off even if requested to be on using other options. This is sometimes necessary for multiple load cases where pressures are applied to different elements in the different load cases.

## MRFOLLO2

Default = 0, SOL 600 only.

Controls whether Marc's follow for, 2 is used when multiple subcases are present.

- 0      Marc's follow for, 2 will not be used when multiple subcases are present. (Default)
- 1      Marc's follow for, 2 will be used when multiple subcases are present

## MRFOLLOW1

No Default, SOL 600 only.

Controls the second field of Marc's follow for, N, M, L parameter (in this case, value N). If entered, options 1, 2, 3 and -1 are currently available. Enter:

- 0      If follower force due to distributed loads (pressure) is not to be considered.
- 1      If follower force stiffness due to distributed loads is not required.
- 2      If follower force stiffness due to distributed loads is to be included.

- 3 If the follower force for distributed loads is based upon the displacement at the beginning of the increment, as opposed to the last iteration.
- 1 If the undeformed geometry is required but total values of distributed loads are to be used (not presently available)

## MRFOLLOW3

Default = 0 for statics and 1 for dynamics.

Controls the 3rd field of Marc's follow for, N, M, L parameter (in this case, value M). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remarks.

### Remarks:

1. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 dynamics and if entered as zero for dynamics will be reset to 1. If the user enters dynamic loads using MARCIN they should be in the form of total loads rather than incremental loads.
2. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 statics if PARAM,MARCTOTL,1 is entered. IF PARAM,MRFOLLOW3 is entered as zero when PARAM,MARCTOTL,1 is entered, it will be reset to 1 if possible.
3. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 if PARAM,MARCTOTT,1 is entered. If PARAM,MRFOLLOW3 is entered as zero when PARAM,MARCTOTT,1 is entered, it will be reset to 1 if possible.

## MRFOLLOW4

SOL 600 only.

Controls the 4th field of Marc's follow for N, M, L parameter (in this case, value L). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remark.

- 0 If follower force for point loads is not required. (Default)
- 1 If follower force for point loads is to be considered.

### Remark:

To activate follower forces (concentrated loads) PARAM,MRFOLLOW,1 must be entered as well as the forces using FORCE1, FORCE2, MOMENT1 and/or MOMENT2. PARAM,MRFOLLOW4 should also be set to one.

## MRGAPUSE

Default = 0, SOL 600 only.

Determines whether Nastran CGAP elements will be approximated as Marc gap elements. The default is to fatal SOL 600 analyses if CGAP elements are found in the model.

- 0      Do not translate Nastran models using Nastran CGAP elements.
- 1      Translate Nastran models using CGAP elements. Marc gap elements are quite different than Nastran elements and usually can't be translated. In a few limited cases the Nastran and Marc gap elements are equivalent. It is up to the user to determine whether the gap Nastran elements can be used with SOL 600 or not. It is suggested that the user read Marc Volume A and C and run small test models to access each particular use of gaps. If gaps can be used, set PARAM,MRGAPUSE,1 and re-run the analysis. See also [MARCAPP](#), [MARGAPN](#), and [MARGAPD](#).

## **MRHERRMN**

Default = 1, SOL 600 only.

This entry controls whether extra grid created for such items as hyperelastic Herrmann elements, CWELD, et.c are output or not in the op2, f06, punch and/or xdb files. When Herrmann grids are output, the displacement value is actually pressure which might be confusing when looking at an f06 file.

- 1      Nodal output for extra grids is not provided.
- 1      Nodal output for extra grids is provided when the above files are requested. (Default)

## **MRHYPMPID**

Default = 0, SOL 600 only.

- 0      Hyperelastic models with mid-side nodes will be translated to Marc including the mid-side nodes. Such models might run quite slowly. (Default)
- 1      Hyperelastic models with mid-side nodes will be translated to Marc leaving out the mid-side nodes. These models will normally run faster, but the displacements of the mid-side nodes will be zero and thus plots might look strange unless the model without mid-side nodes is read into the postprocessor using the .marc.dat or .t16 files.

## **MRITTYPE,**

Default = 0, SOL 600 only.

Type of "constraint" used to connect the auxiliary nodes in all CWELDs.

- 0      RBE3 constraints will be used. (Default)
- 44     Kirchhoff constraints will be used
- 80     Combined RBE2 and RBE3 constraints will be used.

## MRMAT8A3, Value

Default = 1.0, SOL 600 only.

Value      If solid composites are modeled using MAT8, the third coefficient of thermal expansion, ALPHA3 is not defined. ALPHA3 is calculated as ALPHA3=value \* ALPHA1.

## MRMAT8E3, Value

Default = 0.1, SOL 600 only.

Value      If solid composites are modeled using MAT8, the third modulus, E3 is not defined. E3 is calculated as E3=value \* E1.

## MRMAT8N1, Value

Default = 0.5, SOL 600 only.

Value      If solid composites are modeled using MAT8, NU31 is not defined. NU31 will be calculated as NU31=value \* NU12.

## MRMAT8N3, Value

Default = 1.0, SOL 600 only.

Value      If solid composites are modeled using MAT8, the NU23 is not defined. NU23 is calculated as NU23=value \* NU12.

## MRMAXISZ

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as -maxsize N where N is the integer entered.

## MRMAXNUM

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as -maxnum N where N is the integer entered.

## MRMEMSUM

Default = 0, SOL 600 only.

If MRMEMSUM=1, a summary of memory used by the internal Nastran-to-Marc translator will be printed in the f06 file. Each line will contain four numbers (all are in 4-byte words). The first number is the current memory request, the second the current memory (in addition to standard MD Nastran open core), the third is the memory limit with zero meaning no memory limit, and the fourth is the high water memory used so far by the Nastran-to-Marc translator and/or the t160p2 results conversion.

This parameter must be entered in the Case Control.

### **MRMTXKGG, Name**

No Default, SOL 600 only.

Name      Character. The NAME will be used for K2GG entries if entered. This parameter is similar to MRMTXNAM. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

### **MRMTXNAM, Name**

No Default, SOL 600 only.

Name      Character. The NAME (field 2 of the DMIG entry) that will be used for DMIG values in a spawned Nastran execution. For example, a common name used frequently is K2XX. The Case Control command K2PP=Name will be added (in this example K2PP=K2XX will be added) at the end of the Case Control of the spawned job. DMIG entries with other names may exist on the file, but only those with NAME will be used in the spawned execution. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

### **MRNOCOMP**

Default = 1, SOL 600 only.

Determines whether n-layer composite will be created if CORDM is defined on the PSOLID entry. This allows output in material coordinate systems.

- 1      Composite are not created (automatically activated for brake squeal). (Default)
- 1      1-layer composites are created.
- N      N-layer composites are created.

## MRNOCOR

Default = 0, SOL 600 only.

|   |   |
|---|---|
| 0 | Marc parameters ELASTICITY, PLASTICITY, UPDATE, LARGE DISP, CONSTANT DILATATION will be automatically adjusted as recommended by the Marc developers.   |
| 1 | the above parameters will be adjusted as determined to be the most consistent in correlation between Nastran and Marc results for similar problems. This parameter should be entered for buckling problems without plasticity. PARAM,LGDISP,1 should also be entered. |

## MRNOECHO

Default = 0, SOL 600 only.

Determines whether various outputs are placed in the Marc file.

- 0 Output is not suppressed (unless other entries are made to suppress it). (Default)
- 1 Suppress echo of nodes and element lists
- 2 Suppress echo of boundary conditions
- 3 Suppress echo of nurbs data

**Note:** Enter any combination to suppress whatever is desired. To suppress all items, enter 123.

## MRORINTS

Default = 99, SOL 600 only.

Controls orientation type for all solid elements in model (see Marc Volume C documentation, ORIENTATION option).

- 1 Edge 1-2
- 2 Edge 2-3
- 3 Edge 3-4
- 4 Edge 3-1
- 5 Edge 4-1
- 6 XY Plane
- 7 YZ Plane
- 8 ZX Plane
- 9 XU Plane
- 10 YU Plane

- 11 ZU Plane  
 12 UU Plane  
 13 UORIENT Define transformation matrix with orient.f user subroutine  
 14 3D ANIO  
 91 Composite orientations will be Edge 1-2 for shells and 3D ANSO for solids and all non-composites will use option 99. This option provides compatibility with earlier versions and requires that all referenced coordinate systems by composite elements by rectangular.  
 99 All orientations are specified by coordinate systems (CORD2R, CORD2C, etc.). (Default)  
 101-112 Shell elements will have options 1-12, solid element will have option 14. (See Note 2)

**Notes:**

- For solid composites, it is necessary to use the PSOLID entry as well as the PCOMP entry. The PSOLID entry should normally have an entry in field 4 which specifies a CORDi entry to use for the material alignment direction.
- If option 101 is picked, shells will have edge 1-2 and solids will have 3D ANISO. If option 102 is picked, shells will have edge 2-3 and solids will have 3D ANISO, etc.

**MROUTLAY**

Default = -9999, SOL 600 only.

Indicates which shell or composite layers are to be output using the MARCOUT Bulk Data entry or by default. Options are as follows:

- 9999 All layers will be output. (Default)
- N Layers 1 and abs(N) will be output. (N must not be -1.)
- 1 If the number of layers is less than 21, same as -N option. If the number of layers is greater than 21, layers 1, 11, 21, 31, ..., N will be output.
- 0 Top and Bottom layers only will be output.
- N Layers 1 through N will be output.
- 9999 Stresses at the element center only will be output (top and bottom are not output). This option produces output with the assumption that the element has constant stress and strain throughout the element.

**Remarks:**

- Options -9999, -N, -1 and N also produce output at the element center.
- MROUTLAY = 0 is the same as MROUTLAY = -1.
- Do not enter this parameter if Bulk Data entry MLAYOUT is used.

## **MRPARALL**

Default = 0, SOL 600 only.

Determines whether parallel processing for SOL 600 will be forced even if Nastran detects that it might fail prior to spawning Marc.

- 0      Spawn Marc in single-processor mode if Nastran detects Marc may fail in DDM mode.  
(Default)
- 1      Spawn Marc in DDM mode even if Nastran detects Marc may fail. Leave any COORD SYS entries in the Marc input file. If COORD SYS occur in the input file a DDM Marc job may fail depending on the version of Marc being used. Please read the following remark carefully before using this option.
- 2      Spawn Marc in DDM mode even if Nastran detects Marc may fail. Remove all COORD SYS entries from the Marc input file. Please read the following remark carefully before using this option.

### **Remark:**

Known situations where Marc does not presently work in parallel are as follows:

When local coordinate systems are specified for the following:

Field 7 of any GRID entry.

Material coordinate systems (for example field 8 of CQUAD4).

If CBUSH uses coordinate systems.

For the following cases, this situation can be avoided and PARAM,MRPARALL does not need to be entered:

If coordinate systems are used only for input (field 3 of the GRID entry) or just entered in the Nastran input but not actually used, enter PARAM,MRCORDS,1 to allow parallel processing to proceed.

If coordinate systems are only used for shell orientation (such as field 8 of CQUAD4) and if all such coordinate systems are rectangular the following two parameters may be used to allow parallel processing.

PARAM,MRCORDS,1

PARAM,MRORINTS,1

## **MRPBUSHT**

Default = 0, SOL 600 only.

Determines whether the requirement that when PBUSHT and contact occur in the same model, BCTABLE must be specified for each subcase.

- 0      This requirement is enforced and the job will abort if BCTABLE's are not entered
- 1      This requirement will be ignored and the job will proceed. Wrong results may occur and/or the job may terminate later in the run.

## MRPIDCHK

Integer, Default = 1, SOL 600 only.

Controls whether a check for duplicate PIDs (property IDs) will be made in SOL 600.

- 1      Do not make a check for duplicate PIDs.
- 1      Check for duplicate PIDs for structural elements - if any are found fatal the job.

### Remarks:

1. This parameter may be set in RC files.
2. This parameter may be used in the MD Nastran 2010 and subsequent versions.
3. Duplicate PIDs will usually cause SOL 600 to fail or obtain the wrong results if the duplicates occur for structural elements. It is usually acceptable to have a duplicate PID for a spring or damper and for a structural element.
4. For versions prior to MD Nastran 2010, SOL 600 behaved as if PARAM,MRPIDCHK,-1 was set.

## MRPELAST

Default = -1, SOL 600 only.

Determines whether PELAST will be skipped or cause the job to abort for SOL 600. SOL 600 does not support PELAST. PBUSHT along with CBUSH and PBUSH should be used instead.

- 1      SOL 600 jobs with PELAST that are referenced by any CELAS will abort.
- 1      PELAST entries will be skipped (ignored).

## MRPLOAD4

Default = 2, SOL 600 only.

Determines whether continuation entry for PLOAD4 will cause the run to stop or if the continuation line(s) are to be ignored.

- 0      The job will stop and a “Severe Warning” message will be issued.
- 1      The job will continue and the continuation lines will be ignored. A warning message will be issued for the first few such entries.
- 2      CID, N1, N2, N3 and SOLR will be used. If SORL=LINE, the direction must be specified using the CID, N1, N2, N3 fields for SOL 600. LDIR is ignored. (Default)
- 3      If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

### Remarks:

1. MRPLOAD4 should not be set to 2 for axisymmetric or plain strain analyses or if parallel processing is used unless the “single file Marc input file” option is used. In some cases, but not all, if MRPLOAD4=2 is set it will automatically be changed to zero one of these conditions exist.
2. See parameter MRPLOD4R for a companion control.
3. If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

### MRPLOD4R

Integer, Default = 0, SOL 600 only.

Determines how PLOAD4 pressures are treated in Marc when PARAM,MRPLOAD4,2 is set.

- |   |  |
|---|--|
| 0 | Pressures are treated as constant over the element if the pressure over the element varies, the average is applied.      |
| 1 | The Marc PLOAD4 style of loads is used which allows the pressures at the four corners of a quad surface to be different. |

### Remark:

Versions of Marc including MDR3 and Marc 2008R1 and versions previous to these versions produce incorrect results for some models using MRPLOD4R=1 if total loads (specified by PARAM,MARCTOTL,1 or MARCTOTT,1) are used even if the pressure is the same at the four corners of each quad element.

### MRPLSUPD

Default = 1, SOL 600 only.

Determines whether Marc’s PLASTICITY,3 will be used or not for fast integrated composites or smeared composites. Wrong results may be obtained if PLASTICITY,3 is used for these types of analyses even if the plasticity is in non-composite portions of the model (Default = 1 if this parameter is omitted).

- |    |   |
|----|---|
| -1 | Use PLASTICITY,3 if it is necessary for non-composite regions of the model                |
| 1  | Replace PLASTICITY,3 with UPDATE (FINITE and CONSTATN DIALITATION are not used. (Default) |

### Remark:

SOL 600 determines if smeared composites are used by the presence or lack of the SMEAR word on the SOL 600, ID entry. Fast composites are determine if PARAM,MFASTCMP is set to 2 or 3 or if any PCOMPF Bulk Data entries are present in the model.

### MRPOISCK

Integer, Default = 2, SOL 600 only

Controls whether to check if a “bad” Poisson ratio has been entered in SOL 600 for MAT1 entries.

- 0      Do not check or correct “bad” Poisson ratios.
- 1      Check if Poisson ratio is negative or > 0.5, if so change it to 0.333.
- 2      Check if Poisson ratio is negative or > 0.5, if so change MAT1 to MATORT with all three E values, the same as E for MAT1, all three G values the same as G for MAT1 and  
 $v_{12}=v_{23}=v_{31}=0.333$

**Remarks:**

1. Current Marc versions do not allow isotropic materials with Poisson ratios having negative values or values greater than 0.5. Previous Marc versions may have allowed this.
2. “Bad” Poisson ratios may occur if the user specifies E and G and leaves the Poisson field on MAT1 blank.
3. If mrposick=1, the value of G will change. The value of E remains as originally specified.
4. If mrposick=0, the job will error out in the Marc phase of SOL 600.
5. This parameter is available starting the MD Nastran 2010.

**MRPREFER**

Default = 1, SOL 600 only.

Determines to output SOL 600 stresses on the t16 file in the standard Marc coordinate system for the element or the “preferred” (layer) coordinate system when the model contains composite elements.

- 0      Stresses are output in the standard coordinate for the element
- 1      Stresses are output in the “preferred” (layer) coordinate system. (Default)

**Remarks:**

1. The standard coordinate system for solids is usually the basic coordinate system. The standard coordinate system for shells and beams is usually the element coordinate. See Marc documentation volumes A and B for further details.
2. Some versions of Patran cannot plot stresses for MRPREFER=1.

**MRPRSFAC**

Default = 0.0, SOL 600 only.

This parameter is primarily used by Versions of SOL 600 prior to MD Nastran R2 and MSC Nastran 2007 which could not support different pressures at the different corners, pressures applied in directions that are not normal to a surface or edge loads. This parameter is not necessary starting with the MD Nastran R2 and MSC Nastran 2007 Version 5 if PARAM,MRPLLOAD4,2 is set.

Factor by which pressure loads are scaled for SOL 600. Prior to MD Nastran 2006 R1 and MSC Nastran 2006, Marc was not capable of handling different pressure applied to different corners of a surface. In other words, different values of p1, p2, p3, p4 on the PLOAD4 entry could not be handled. Approximations can be made by assuming the pressure is uniform over the surface if the surface is small enough. For SOL 600, the pressure (P) used by Marc is calculated using the following:

If p1 is not blank (or zero) regardless of the value of mrprsfac  
 $P = p1$

If mrprsfac is positive and p1 is blank or zero and one or more of p2, p3, or p4 are not zero or blank,  
 $P=mrprsfac*(p1+p2+p3+p4)$  (This default if p2 or p3 or p4 are not zero or blank)

If mrprsfac is zero or negative,

If none of p1, p2, p3, p4 are blank or zero  $P=0.25*(p1+p2+p3+p4)$

Otherwise,

$P=(+/-)pp$  where pp is the value largest absolute value of p1, p2, p3, p4 and P will have the proper associated sign.

### Remarks:

1. If MRPLOAD4=2, MRPRSFAC is ignored and p1,p2,p3,p4 are used as entered.
2. If MRPLOAD4 is not 2, the default for mrprsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrprsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by PLOAD4.

**Note:** The default for mrpsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrpsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by pload4.

## MRPSHCMP

Integer, Default = 0, SOL 600 only

If MRPSHELL=1 and there is a Marc PSHELL and composite for the same elements, this option controls which will be used.

- 0      Use Marc composite and ignore Marc pshell. (Default)
- 1      Use Marc pshell and ignore Marc composite.
- 2      Do not make any corrections (this works for some models and fails for others)

## MRPSHELL

Default = 0, SOL 600 only.

Used to control shell property specifications for Marc in SOL 600.

- 0      Classical GEOMETRY options will be used for all shell elements. (Default)  
 1      The Marc PSHELL option will be used, (can only be used with CQUAD4, CQUAD8, CTRIA3, and CTRIA6).

**Important:** If MRPSHELL=1, shell elements will use Marc's new PSHELL option. No materials used by PSHELL may be referenced by other types of elements that this option has used. For commonly used elements, duplicate materials will automatically be created by SOL 600 to satisfy this criteria, however this can take considerable computer time. If the user knows that no materials are used by both PSHELL and other properties, he can set PARAM,MRDUPMAT,-1 to bypass the checking and creation for duplicate materials. To speed up this process, set PARAM,MSPEEDPS,1. The Marc PSHELL formulation is more stable if MID3=MID2. To set MID3=MID2, set PARAM,MARCMID3,1.

## MRRBE3TR

Default = 0, SOL 600 only.

Controls whether all translations or all translation and rotations will be used for the reference degrees of freedom or whether fewer than 3 are allowed.

- 0      Fewer then 3 reference degrees-of-freedom may be used. (Default)  
 3      All RBE3's with REF C (field 5 of the RBE3 entry) not having 123 will have all translations for the Marc REFC (field 5 of the Marc RBE3 entry) set to 123.  
 6      All RBE3's with REF C (field 5 of the Nastran RBE3 entry) not having 123456 will have all translations and rotationf or the Marc REFC (field 5 of the Marc RBE3 entry) set to 123456.

### Remarks:

1. Marc frequently gives a singular tying message if there are not at least degrees of freedom 123 for REFC. This sometimes results in poor singularity ratios and incorrect results for nonlinear analyses. It also can prevent eigenvalues from being calculated.
2. It is recommended that option 3 or 6 be used for all SOL 600 analyses unless the user is certain that REFC can have fewer then three dof's for the particular analysis.

## MRRCFILe, RCF

No Default, SOL 600 only.

- RCF      Character. Name of RCF file name (limited to 8 characters) used in conjunction with another MSC Nastran run spawned from an original MSC Nastran run as specified by the CONTINUE option on the SOL 600 command. The RCF file may contain any information required (such as scratch=yes, exe=, etc.) as discussed in [Executing MSC Nastran, 2](#) of this guide. This rcf file does not have to use the same options as the primary rcf file and should normally set batch=no as one of the options.

## MRRELNOD

Default = -1, SOL 600 only.

Controls enforced displacements using SPCD.

- 0      For multiple subcases with SPCDs, the SPCDs from the previous subcase will be released at the start of the current subcase.
- 1      SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be released gradually during the current subcase.
- 1     SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be subtracted out. For example, if the previous subcase applied a SPCD of 0.1 to a particular dof, the current subcase will apply -1.0. This will bring the displacement of that dof to zero for the current subcase.

## MRRSTOP2

Default = 0, SOL 600 only

Integer, Normally op2, xdb, punch and f06 output is not available for SOL 600 restart analyses. Setting this parameter to 1 will allow the program to attempt to create one or more of these files. Only in limited cases will the job be successful.

- 0      Do not attempt to create .op2, xdb, punch, f06 output for restart runs (default if parameter is not entered). (Default)
- 1      Attempt to create .op2, xdb, punch, f06 output as specified by other options.

## MRSCMOD

Default = 0.0, SOL 600 only.

Solution scaling factor for linear buckling analysis (SOLs 600, 105) using the Lanczos method. If the applied load in the first subcase is too large, the Lanczos procedure may fail. This number may be used to scale the solution for numerical reasons. The collapse load will be output based upon the total load applied.

### Remark:

This parameter applies to rigid surfaces described using 4-node patches only.

## MRSETNA1, N

Default = Program calculated, SOL 600 only.

- N        If this parameter is entered with N>0, the integer value entered will be used in Marc's SETNAME parameter section as the first value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

## MRSETNA2, M

Default = Program calculated, SOL 600 only.

- M        If this parameter is entered with M>0, the integer value entered will be used in Marc's SETNAME parameter section as the second value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

## MRSETNAM, N

Integer, Default = Program calculated, SOL 600 only.

- N        If this parameter is entered with N>0, the value entered will be used in Marc's SETNAME parameter section. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. To get around a Marc bug, if a computed setname value is large, Nastran will normally use the undocumented form (see MRSETNA1). To prevent this, set N to -1.

## MRSPAWN2, CMD

Default = Nastran, SOL 600 only.

CMD      Character. Name a command to run D Nastran (limited to 8 characters single field or 16 for large field) used in conjunction with the CONTINUE options on the SOL 600 command. The Nastran run to be spawned will have the form:

CMD jid.nast.dat rcf=RCF

where file RCF is provided by PARAM,MRRCFILE,RCF. As an example, if CMD is nastran, jid is myjob (original file myjob.dat) and RCF=nast.rc, the spawned run will execute using:

nastran myjob.nast.dat rcf=nast.rc

### Notes:

1. See PARAM\*,HEATCMD for SOL 600 thermal contact heat transfer analysis.
2. CMD will be converted to lower case regardless of the case entered.

## MRSPLINE

Default = 2, SOL 600 only.

Determines whether RSPLINE will be converted to CBAR or to MPC's.

- 1      RSPLINE will ignored (this option should only be used for model checkout)
- 1      RSPLINE will be converted to stiff CBAR elements
- 2      RSPLINE will be converted to MPC's (Marc's servo link) using the same equations used by other Nastran solutions, except that the MPC's are capable of large rotation. This option requires somewhat longer translation times than the others but can save computer time during the Marc execution phase. (Default)

## MRSPRING

Default = 0.0, SOL 600 only.

Specifies a stiffness value to be added to the main diagonal of each translational term of the stiffness matrix. This option is useful in nonlinear static analysis with 3D contact of two or more separate structures. Some of the pieces may not be grounded until contact occurs. By adding a small spring to ground, such as K=1.0, these pieces are stabilized until contact occurs. This option applies to SOL 600 only and generates SPRINGS in the Marc input file for all nodes in the model and all three translational degrees-of-freedom. IF K is entered as a negative number, the absolute value of K will be added to all 6 degrees of freedom of each grid in the model.

If the run is for heat transfer and K is positive, the spring will only be added to DOF 1.

## MRSRVER

Integer, Default = -1, SOL 600 only

Controls how CELAS and all other items map to the Marc input data.

- 0 Non-table form is used which generally means the Marc input file version is less than 11.
- 11 The table form for SPRINGS is used which requires two lines, the first with spring rates and the second with table ID's. PARAM,MRSRVER,11 is usually required if PARAM,MRCONVER,11 is entered.

## MRSTEADY

Default = 1, SOL 600 only.

Controls the solution method for SOL 600 steady state heat transfer.

- 1 Marc STEADY STATE is used with TIME STEP of 1.0. (Default)
- 2 AUTO STEP is used.

## MRT16STP, N

Default = 0, SOL 600 only.

Enter in Case Control at the subcase level.

- N Number of load increments to put on the t16/t19 files for each subcase when the AUTO STEP method is used. The default of 0 puts all adaptive increments on the t16/t19 file. If a value of N is entered, load steps for times=tmax/N will always be introduced into the auto stepping process and the t16/t19 files will have outputs only at zero and those times. This corresponds to field 1 of Marc's AUTO STEP 2nd option and can also be set using the NLAUTO option. If this is the only non-default NLAUTO variable to set, it is more easily accomplished using this parameter. It is suggested that this parameter always be used for large models and that N be 10 or greater, otherwise the size of the t16/t19 files may become very large. This is especially important for Windows systems, which presently has a 4GB limit in converting t16 to op2 files due to compiler limitations.

## MRTABLIM

Integer, Default = 0, SOL 600 only

Determines the maximum number of points in any TABLES1 used when tables are specified or MATHE input.

- 1 Unlimited, not recommended - If used could lead to numerical errors or program aborts
- 0 The limit is 500 points.
- N The limit is set to N points. It is recommended that N does not exceed 500.

**Remark:**

If more than about 500 points are used for the tables, the curve filling processes used to determine the material constants may encounter numerical errors and the program could abort. If a table with more than the number points specified by this parameter is entered, the number of points will be reduced to the specified limit so that numerical errors are unlikely to occur.

**MRTABLS1**

Default = 0, SOL 600 only.

TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves according to the following formulas for the value entered for MRTABLS1. In the formulas s is the stress entered for TABLES1, e is the strain entered in TABLES1, S is the Marc WORK HARD stress and E is the Marc WORK HARD Plastic strain. ey is the yield strain ( $sy/E$ ) where EE is Young's modulus, sy is the yield stress. The first point of the Nastran curve will be skipped unless MRTABLS2 is set to 1.

|             |                                      |
|-------------|--------------------------------------|
| 0 (Default) | $S=s$<br>$E=e - ey$                  |
| 1           | $S=s(1+e)$<br>$E=\ln(e+1)$           |
| 2           | $S=s$<br>$E=e - s/E$                 |
| 3           | $S=s(1+e)$<br>$E=\ln(1+e) - s/EE$    |
| 4           | $S=s$<br>$E=e$                       |
| 5           | $S=s$<br>$E=\ln(1+e)$                |
| 6           | $S=s$<br>$E=\ln(1+e-s/E)$            |
| 7           | $S=s(1+e)$<br>$E=\ln(1+e)-S/EE$      |
| 8           | $S=s(1+E)$<br>$E=\ln(1+e)$           |
| 9           | $S=s(1+E)$<br>$E=\ln(1+e)-\ln(1+ey)$ |

**Note:** This parameter (like any other parameter) can only be entered once in an input file.

## MRTABLS2

Default = 0, SOL 600 only.

- 0 TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves starting with the yield point. The first point will be skipped. (Default)
- 1 All points in TABLES1 will be connected to WORK HARD, however, the first plastic strain will be set to zero if entered as non-zero.

**Note:** This parameter (like any other parameter) can only be entered once in an input file.

## MRTABPRC

Integer, Default = 0, SOL 600 only

Determines the precision of stress-strain curves when translated from Nastran to Marc using SOL 600.

- 0 Standard precision (8 significant figures) is used.
- 1 Maximum precision (14 significant figures) is used.

### Remarks:

1. This parameter is available starting with the 2010 entry. For MD Nastran 2010 it only applies to stress-strain curves entered in conjunction with MATEP.
2. Prior to MD Nastran 2010, only option zero was available.
3. In most cases, the stress-strain values are not known to even 8 significant figures. However, if the stress-strain is calculated from other data the extra precision might be useful.

## MRTFINAL

SOL 600 only.

Real. In some dynamic problems, due to numerical roundoff an extra time step with very small initial and final times is generated. If these times are too small, Marc sometimes will diverge even though the “final time” actually desired by the user has been reached within reasonable accuracy. MRTFINAL is a value below which this extra step will be eliminated to prevent excess computations and/or possible divergence. If this parameter is not entered, the default value is 1.0E-8. This parameter is only used with the AUTO STEP procedure. The ‘extra’ step will also be eliminated if the initial time step is less than MRTFINAL/100.0.

## MRTHREAD

Integer > 0, Default = 1, SOL 600 only.

Determines the number of threads to be used by the run. For multi-core processors speed can frequently be gained by specifying a number of threads equal to the number of cores assuming the job has exclusive use of the machine.



- 1        Use 1 thread.
- 2        Use 2 threads.
- N        Use N threads.

### Remarks:

1. This parameter may only be used by solver types 8 (multifrontal direct sparse) and 11 (Pardiso).
2. This parameter may be combined with PARAM,MARPROCS to use several processors each using several threads.

### MRTIMING

Default = 0, SOL 600 only.

If MRTIMING is 1, timing summaries for various portions of the internal Marc translator will be provided in the f06 and log files.

### MRTSHEAR

Default = 0, SOL 600 only.

By default, parabolic shear deformation is not included in the formulation of beam and shell elements if there are no composite elements. If composite shell elements are present, parabolic shear deformation is included by default. This parameter can be used to override these defaults.

- 1        Parabolic shear deformation is included in the formulation of beam and shell elements.
- 1 or 0    Parabolic shear deformation is not included even if composite elements are present.  
(Default)

### Remark:

This parameter maps to Marc TSHEAR parameter.

### MRUSDATA

Default = 0, SOL 600 only.

Determines whether user subroutine USDATA will be activated and if so the size of data.

- 0        User subroutine USDATA is not active in the current run. (Default)
- N        User subroutine USDATA will be active the in the current run and N is the number of real\*4 words needed for the data stored in common block /usdam/

### Remark:

In addition to adding this parameter, the user must enter USDATA on Bulk Data entry USRSUB6.

## MRVFIMPL

Default = 0.01, SOL 600 only.

Controls the fraction of the maximum view factor that is to be treated implicitly (contribute to operator matrix). View factor values smaller than this cutoff are treated explicitly.

## MRV09V11

Default = 1, SOL 600 only.

Determines whether certain Marc “features” which are default in Marc “version 11” are added for SOL 600-generated models that use “version 9”.

- 1      Do not add the features.
- 1      Add the following features:
  - feature,4703 to speed up DDM jobs for one-processor jobs, it has no effect
  - feature,5701 to disable old rigid rotation checking which was too stringent
  - feature,601 to improve contact
  - feature,5301 to improve deformable-deformable contact
  - feature,3201 to improve contact friction types 6 and 7“
  - feature,5601 to improve thickness updating when the updated Lagrange method is used
  - feature5801 to improve in-plane bending of advance nonlinear element type 140
  - feature,6001 to improve concrete cracking analysis

**Note:** The above features are only used for certain problems, even though all are included with the default option, they have no effect on models that do not use advance nonlinear element type 140, feature 601 has no effect on models that do not have contact, etc.

## MSIZOVRD

Default = -1, for small models and +1 for large models, SOL 600 only.

Determines whether Marc SIZING values for large models will be updated or not. Nastran includes Marc directories with a file named “include” in the tools directory which includes a line MAXNUM=N, where N is some value like 1000000. If the maximum number of nodes or elements in the model exceeds N, memory overwrites or job aborts are possible unless either N is set larger than the actual max node or element number in the model or the values are specified on the SIZING entry (field 3 for max element and field 4 for max node).

- 1      The sizing entry will not be updated (either the model is not large, MAXNUM in the include file has been updated or parameters such as MARCSIZ3 and/or MARCSIZ4 have been entered into the run to provide values that are large enough. (Default)
- 1      If the number of nodes or elements in the model exceeds 1,000,000 the sizing entry will be updated to the max number of nodes and elements actually in the model. Extra nodes and/or elements to account for welds, pinned members, Herrmann elements, etc. will be included.

**Note:** For PARAM,MSIZOVRD large models are considered to be those with more than 1,000,000 nodes and/or elements.

## MSOLMEM, MBYTE

Default = Program determined value, SOL 600 only.

MBYTE      If entered, the integer value entered here is the 8th field of Marc's SOLVER option, and is the maximum memory in Mega Bytes that can be used by Marc's solver types 6 (hardware provided direct) and 8 (multi-frontal direct, which is the default solver for SOL 600) before going out of core. This parameter is the same as the MBYTE field on the NLSTRAT entry any may be easier to enter as a parameter if no other NLSTRAT values are needed.

**Note:** This parameter should only be entered if NLSTRAT entries are not required. If any NLSTRAT entries are made, use the MBYTE field instead of this parameter.

## MSPCCCHK

Integer, Default = 0, SOL 600 only

Effects multiple subcases where SPCD's are applied and SPC's potentially change between the subcases.

- 0      A comprehensive check is made to ensure that all SPC changes are accounted for all subcases.
- 1      The check is not made. The check can take significant computer time and in most cases it is not actually needed. If the user knows it is not needed, computer time can be saved by turning it off.

### Remarks:

1. This parameter is available starting with MD Nastran 2010.
2. This parameter may be set in rc files.

## MSPCKEEP

Default = 0, SOL 600 only.

Determines which SPC's entered in the bdf file are retained in the Marc input file for 2D models.

- 0      Only retain those dof's that are normally associated with the type of 2D analysis (normally 2 translations) (Default)
- 1      Retain all dof entered in the bdf file in the Marc input file.

### Remark:

Some types of Marc 2D analyses support one rotation in addition to two translations and it is important to restrain this rotation.

## MSPEEDCB

Default = -1, SOL 600 only.

Determines whether CBEAM increased speed options are to be applied. This option may be necessary for models with a large number of beams whose element IDs are large.

- 1 No increased speed options are used for CBEAM. (Default)
- 1 cbeam.prp file will be standard unformatted rather than direct access. A small table to provide the location of each entry is saved in memory.
- 2 The entire cbeam.prp file is saved in memory

**Note:** See param,marccbar,1 to change CBAR to CBEAM.

## MSPEEDCW

Default = 0, SOL 600 only.

Determines whether CWELD elements will be translated to Marc in core (for increased speed) or out of core. This parameter is needed if many CWELD elements are present in the model to avoid large translation times.

- 0 Processed out of core. (Default)
- 1 Process in core.

## MSPEEDOU

Default = 1, SOL 600 only.

Determines whether speed enhancements are activated for the t16op2 conversion.

- 0 Speed enhancements are not activated.
- 1 Speed enhancements are activated which will place certain scratch data in memory. For large models make sure enough memory is available (if PARAM,MSPEEDSE and/or PARAM,MSPEEDP4 are used, there is probably sufficient memory unless there is a large number of output "time" points. (Default)

## MSPEEDP4

Default = -1, SOL 600 only.

Determines whether PLOAD4 entries will be translated to Marc in core (for increased speed) or out of core.

- 1 PLOAD4 will be processed in core.
- 1 PLOAD4 will be processed out of core. (Default)

**Note:** The MSPEEDP4=1 option may require more memory than is available on certain computers for large models. Do not use if CWELD elements are present.

## MSPEEDPS

Default = -1, SOL 600 only.

Determines whether additional memory will be used if the Marc PSHELL option is invoked (PARAM,MRPSHELL,1) or smear option on SOL 600, ID in order to speed up processing. This will be beneficial if there are many materials and properties in the model and not very beneficial if all elements use just a few properties and materials.

- 1 No additional memory will be used. (Default)
- 1 Additional memory will be used if available.

## MSPEEDSE

Default = -1, SOL 600 only.

Determines whether speed enhancements are activated using extra memory and/or special low level I/O routines.

- 1 No speed enhancements will be activated. (Default)
- 1 Solid elements 2D elements data will be processed in core.
- 2 All elements will be processed in core.
- 3 2D and 3D elements will be processed in core, 1D elements will be processed using bioxxx (low level direct access routines used by gino).

**Note:** The MSPEEDSE=1 option may require more memory than is available on certain computers for large models. MSPEEDSE > 0 should not be used for 64-bit integer versions of Nastran or DEC Alpha computer systems.

## MSPEEDS2

Default = -1, SOL 600 only.

Determines whether additional memory will be used when PARAM,MSPEEDSE with options 1, 2, 3 is entered to obtain additional speed.

- 1 Additional memory will not be used. (Default)
- 1 Additional memory will be used and additional speed will usually be obtained.

## MSPEEDSH

Default = -1, SOL 600 only.

Determines whether additional speed enhancements will be used to process shell/plate elements (CQUAD4, CTRIA3, etc.) using incore memory.

- 1 No additional speed enhancements are used. (Default)
- 1 Additional speed in-core enhancements will be used. Sufficient memory must be available. If not, the job will abort with an appropriate message.

**Note:** This parameter should only be used if shell speed enhancements are selected with one of the PARAM,MSPEEDSE options.

## MSPLINCO

Integer, Default = 0, SOL 600. For SOL 400, MSPLINCO =1 is always used.

This parameter controls whether to enforce C0 continuity for all spline options if any are requested by setting IDSPL=1 on any BCBODY entry.

- 0 C0 continuity is not enforced.
- 1 C0 continuity is enforced.

## MSTFBEM

Default = 0, SOL 600 only.

Determines if all rigid elements (RBE2, RBE3, RBAR, RTRPLT) will be converted to stiff beams. This parameter should only be used if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 1 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. All 6 dof's at each end of the rigid beam will be used.
- 2 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. Pin codes specified by the original Nastran rigid elements will be used when possible.

### Remark:

See PARAM,MARCSCLR to specify a scale factor for the "Default" properties of these stiff beams.

## MSTRFREE

Integer, Default = 0

Determines whether the nodal coordinates in the GEOM1 datablock of SOL 600 OOP2 files will have the original coordinates or revised stress-free coordinates.

- 0 GEOM1 will have the original coordinates (same as defined on GRId entries)
- 1 GEOM1 will have modified coordinates to enforce stress-free conditions if applicable

### **Remarks:**

1. ICOORD on the BCTABLE with ID=0 entry must be set to 1 or 3 for at least one slave-master combination.
2. The parameter only addresses op2 files (t16 and t19 files automatically use MSTRFREE=1)
3. This capability is available starting with MD Nastran 2010. Prior version always used MSTRFREE=0.

## **MSTTDYNE**

Integer, Default = 0, SOL 600 only

Controls whether SOL 600 may have static and dynamic load cases in the same analysis.

- 0 All subcases must either be static or dynamic
- 1 Mixed static and dynamic load cases are allowed.

### **Remarks:**

1. Use of this entry requires SOL 600,106 or SOL 600,NLSTATIC (not SOL 600,NLTRAN).
2. Do not enter , DLOAD, LOADSET, LSEQ, TLOADi in the Case Control or Bulk Data.
3. Each subcase must have NLAUTO entries. Whether the subcase is static or dynamic is controlled by IFLAG on each NLAUTO entry. For static subcases IFLAG=3 and for dynamic subcases IFLAG=0.
4. All loads in each subcase build up from an initial value at the end of the previous subcase to the full value entered on the various load-type entries (such as FORCE, PLOAD4, SPCD, etc.) at the end of the subcase. Time histories for individual load cases are not allowed with this parameter. If all loads for a particular subcase use the same time history, this subcase may be divided into N-1 smaller subcases where N is the number of points on the time history curve. For this situation, the loads for each smaller subcase should be scaled by the amplitude of each point on the time history curve. This technique may also be used to simulate force vs. time for an analysis with all static subcases.
5. When MSTTDYNE=1, the following will automatically be added to the Bulk Data:

```
marcin,-1,0,dynamic,6
param,marcauv,0
```

These two entries may be added by the user in place of PARAM,MSTTDYNE,1 however both should not be done.

## **MTABIGNR**

Default = 0, SOL 600 only.

Determines whether tables for VCCT analyses will be ignored or used.

- 1 VCCT tables are not used
- 0 VCCT tables are used. (Default)

**Remark:**

Use of tables for VCCT SOL 600 2D analyses with contact is not supported. If the table variation is not important, this parameter may be used to obtain an approximate solution.

**MTABLD1M**

Default = 1, SOL 600 only.

Option to modify or not modify all TABLED1 entries which do not start with the first point of (0.0, 0.0).

- 1 Do not modify any TABLED1 entries
- 1 Modify all TABLD1 entries that do not start with (0.0, 0.0). (Default)

**Remark:**

See PARAM,MTABLD1T.

**MTABLD1T**

Default = 0.01, SOL 600 only.

Specifies the second time value of all TABLED1 entries that do not start with the first point being (0.0, 0.0) if PARAM,MTABLD1M=1. Modified tables will start with the first point of (0.0, 0.0), the second point will be at the time specified by this parameter with an amplitude of the original first amplitude. The 3<sup>rd</sup> point will be at time 2\*MTABLD1T and amplitude of the original first amplitude.

**Remark:**

The proper value of MTABLD1T to enter to analyze a step input depends on the fundamental natural frequency of the model. If MTABLD1T is too small, the response will not fully build up and if it is too large, it will act like a ramp input rather than a step. The best value to use is approximately 0.1/f<sub>n</sub>, where f<sub>n</sub> is the first linear natural frequency in Hz.

**MTEMPCHK**

Integer, Default = 0, SOL 600 only

Controls how temperature-dependent properties are checked in the Marc portion of SOL 600.

- 0 Check at zero degrees only. (Marc feature 9601)
- 1 Check at all temperatures, stop checking and fatal the job when the first problem is found. (Marc feature 9600)
- 2 Check at all temperatures and report all problems, then fatal the job if any problems were found. (Marc feature 9602)

**Remarks:**

1. This parameter is available starting with MD Nastran 2010.
2. The default in Marc previously was feature 9601.
3. The default in SOL 600 is different than in Marc. The default in Marc is 9600.
4. Checking in Marc will occur from the lowest temperature of any curve to the highest of any curve. Thus, to ensure that all curves pass, define all curves from the lowest temperature of any curve to the highest temperature of any curve (even though one particular curve can never reach the lowest or highest temperature specified on another curve).
5. features,960i will be entered if any TABLEMi or TABL3Di are found.

**MTEMPDWN**

Default = -1, SOL 600 only.

Option to automatically choose the FeFp multiplicative decomposition plasticity model (PARAM,MARCPLAS,5) for plasticity problems with thermal loading when the temperature decreases (see PARAM,MARCPLAS).

- 1      Use the standard mean normal with large strain method unless PARAM,MARCPLAS is entered.  
(Default)
- 1      Attempt to determine whether the temperature at the start of the run is lower than the initial temperature, and if so choose the FeFp method.

**Remarks:**

1. For some problems with very soft materials, the FeFp method is required to achieve convergence for plasticity cases where the temperature decreases during the early stages of the run. Since most users would not know this, PARAM,MTEMPDWN is offered to automatically let Nastran decide to invoke this method or not.
2. The FeFp method takes substantially more computer time than the default mean normal method.
3. This parameter can be entered in RC files if it is desired to be used for a large number of runs.

**MTET4HYP**

Integer, Default = 0, SOL 600 only

Controls settings for TET4 elements with hyperelasticity.

- 0      Use Marc element 157 which uses the Herrmann formulation.
- 1      Use Marc element 134 and automatically set PARAM,MARUPDAT,1

**Remarks:**

1. MTET4HYP=1 should normally only be set if any tet4 elements use MATHE or MATHP and if one or more of the grids in such tet4 have field 7 set to a cylindrical or spherical coordinate system.
2. If Remark 1 does not apply, MTET4HYP=0 is more accurate, although it is highly recommended that tet10 elements be used for all SOL 600 models which will be even more accurate.

## MTLD1SID

Integer, Default = -1 for SOL 600

Determines whether the SID of TLOAD1s entered in the input will be changed or not. (SOL 600 only)

- |    |  |
|----|--|
| -1 | Do not change the SID of any TLOAD1 entry  |
| 0  | Change the SID of all TLOAD1s to match the lowest SID of any TLOAD1 found in the original input. |
| N  | Change the SID of all TLOAD1s to the value N.  |

### **Remark:**

This parameter may only be set in the input file.

## MUALLUDS

Integer, Default = 0, SOL 600 only

Controls how material, contact and element-related user subroutines are specified in SOL 600.

- |   |   |
|---|---|
| 0 | User subroutines are entered using the MATUSR entry (material), contact-related (BCBODY, BCTABLE, etc.) or element entries as well as the USRSUB6 entry.              |
| 1 | User subroutines are entered using the MATUDS, BCONDUDS and/or ELEMUDS. They should not be specified on MATUSR, BCxxx, element entries or by using the USRSUB6 entry. |

### **Remark:**

For version prior to MD Nastran 2010 only option 0 was available.

## MULRFORC

Default = -1, SOL 600 only.

Option to activate multiple RFORCE entries for different portions of The model in the same subcase.

- |    |  |
|----|--|
| -1 | Do not allow this capability (only one RFORCE entry for subcase can be entered). (Default) |
| 1  | Allow multiple RFORCE entries for each subcase   |

## MUMPSOLV

Integer > 0, Default = 1, SOL 600 only.

This can also be set by the Bulk Data entry PARAMARC.

- |   |                  |
|---|------------------|
| 1 | Use 1 processor. |
| 2 | Use 2 processor. |
| N | Use N processor. |

### Remarks:

1. This parameter may only be used by solver the MUMPS solver (solver type 12).
2. The PARAMARC Bulk Data entry cannot specify this type of parallel processing.

### MUSBKEEP

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | If user subroutines are included, they are compiled and linked to form a new version of Marc if MUSBKEEP=0, the new version of Marc will be deleted at the end of the run. (Default) |
| 1 | The new Marc executable will be saved on disk in the same directory as the Nastran input file. Its name will be the name used in PARAM,MARCUSUB with the extension marc.             |

### MVERMOON

Integer, Default = 0, SOL 600 only

Controls whether 5-term Mooney series or 5-constant Mooney will be used in the Marc portion of SOL 600.

- |   |  |
|---|--|
| 0 | 5-constant Mooney formulation will be used.    |
| 1 | 5-term series Mooney formulation will be used. |

### Remark:

Prior to MD Nastran 2010, the Marc only allowed a 5-constant Mooney formulation with no volumetric terms. Now Marc allows a full 5-term series for both distortional and volumetric behavior for both MATHP and MATHE entries. If a version of Marc prior to this is to be used, set param,mvermoon,0 or use the default.

### MWINQUOT

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 run on Windows will attempt to place quote strings in the command to execute Marc when PATH is not specified on the SOL 600 Executive Control statement.

- |   |   |
|---|---|
| 0 | Attempt to place quote strings around various fields. |
| 1 | Do not place any quote strings.                       |

### Remarks:

1. Quotes are normally only required if spaces are needed in the file names or directory names.
2. This parameter may be placed in rc files.
3. Depending on the complexity of the command to execute Marc, quote strings may sometimes be incorrect. If the command is complex, it is suggested that PATH=1 be placed on the SOL 600 Executive statement and the statement to execute Marc be placed in the marcrun.pth file.

### MWLDSGSA

Integer, Default = 1, SOL 600.

Controls CWELD type ELEMID GS, GA, GB usage in SOL 600.

- 0 When GS, GA and GB are all greater than zero for CWELD type ELEMID, Marc ignores GS and uses GA and GB to determine if a connection is possible. MWLDGSA=0 retains this default behavior, however experience shows that some models will fail with a message that the weld connection could not be achieved.
- 1 This option will re-set GA and GB to zero so that GS will be used to achieve the connection. Experience shows that this option may be more reliable for many models with CWELD type ELEMID.

## **MXICODE0**

Default = 5

In nonlinear harmonic response analysis, if the solution fails to converge more than MXICODE0 times in succession, a new trial displacement vector is calculated.

## **MXLAGM1**

Default = 0.0

MXLAGM1 is to override the maximum Lagrange Multiplier that is used by the merit function in Trust Region.

## **NASPRT**

Default = 0

NASPRT specifies how often data recovery is performed and printed in SOL 200. By default, SOL 200, in addition to performing an analysis and optimization, will perform full data recovery operations at the first design cycle and upon completion of the last design cycle.

If NASPRT > 0, then data recovery operations are performed at the first design cycle; at every design cycle that is a multiple of NASPRT; and the last design cycle. For example, if PARAM,NASPRT,2 and the maximum number of design iterations requested is 5, then data recovery is obtained at design iterations 1, 2, 4, and 5.

If NASPRT < 0, then no data recovery operations are performed.

## NDAMP, NDAMPM

### NDAMP:

Default = 0.01 for SOLs 129 and 159, -0.05 for SOL 400 heat transfer and mechanical non-contact analysis, 0.0 for SOL 400 contact.

### NDAMPM:

Default = 0.0 for SOL 400 non-contact, 1.0 for SOL 400 contact. Not used for SOL 400 heat transfer.

### NDAMP/NDAMPM:

In SOLs 129 and 159, numerical damping may be specified for the METHODS = "ADAPT" on the entry through the NDAMP entry in order to achieve numerical stability. A value of zero for NDAMP requests no numerical damping. The recommended range of NDAMP values is from 0.0 to 0.1.

SOL 400 Mechanical: In SOL 400, numerical damping may be specified for all METHODS (ADAPT, FNT, etc. on the entry. NDAMP and NDAMPM are two parameters that control the damping scheme and the associated dynamic operator. The equilibrium equation for the most general operator (the Generalized- $\alpha$  operator) is given by

$$M\ddot{u}_{n+1+\alpha_m} + C\dot{u}_{n+1+\alpha_f} + F_{n+1+\alpha_f}^{int} = F_{n+1+\alpha_f}^{ext}$$

where  $\alpha_m$  is NDAMPM and  $\alpha_f$  is NDAMP. In the previous equation, a typical quantity  $F_{n+1+\alpha}$  is given by the expanded form

$$F_{n+1+\alpha} = (1 + \alpha)F_{n+1} - \alpha F_n$$

Depending on the values of NDAMP and NDAMPM, the equilibrium equations can reduce to the HHT- $\alpha$  scheme (NDAMPM = 0) or the WBZ- $\alpha$  scheme (NDAMP = 0) or the Generalized- $\alpha$  scheme (NDAMPM  $\neq$  0, NDAMP  $\neq$  0). For the HHT- $\alpha$  scheme, NDAMP can vary in the range of  $-0.33 \leq NDAMP \leq 0.0$ . For the WBZ- $\alpha$  scheme, NDAMPM can vary in the range of  $0.0 \leq NDAMPM \leq 1.0$ . For the Generalized- $\alpha$  scheme, NDAMP can vary in the range of  $-0.5 \leq NDAMP \leq 0.0$  and NDAMPM in the range of  $-0.5 \leq NDAMPM \leq 1.0$ .

For problems involving no contact, the HHT- $\alpha$  is used in SOL 400 with default NDAMP = -0.05 and NDAMPM = 0.0 except that the model is linear, or no mass and no damping matrix. In this case, the defaults are NDAMP = 0.0 and NDAMPM = 0.0. For problems involving contact, the WBZ- $\alpha$  scheme is used in SOL 400 with default NDAMPM = 1.0 and NDAMP = 0.0

SOL 400 Heat Transfer: Numerical damping may be specified through the value of NDAMP only. NDAMPM is not used. NDAMP can be varied in the range of (-2.414, 0.414). At these outer limits, the transient scheme reduces to the Backward-Euler method. Any value that is outside this range is automatically reset to the closest outer limit. For NDAMP = 0.0, the transient scheme reduces to the Crank-Nicholson scheme. The default value of NDAMP is -0.05.

SOL 400 Thermo-Mechanical Coupled Analysis: If NDAMP and NDAMPM are unspecified, then the default that is appropriate for the particular sub-step is used. For e.g., for a coupled transient contact problem, NDAMP = -0.05 during the heat transfer sub-step and NDAMP = 0.0, NDAMPM = 1.0 during the mechanical sub-step. If the values of NDAMP and NDAMPM are specified by the user in the Bulk Data

Section, then these values would be applicable for both the thermal and mechanical sub-steps. To make the values of the NDAMP/NDAMPM physics-dependent, the user can define the values under each sub-step of the Case Control Section. E.g.,

SUBSTEP 1  
ANALYSIS=HTRAN  
PARAM,NDAMP,-2.414

SUBSTEP 2  
ANALYSIS=NLTTRAN  
PARAM,NDAMP,-0.05  
PARAM,NDAMPM,0.0

## NEWMARK

Default = NO

See CQC under [SCRSPEC, 987](#).

## NFDOPTS

Default = 2

PARAM, NFDOPTS, 2 computes frequency dependent matrices at master frequencies. In general, master frequencies via MFREQ in case control section of input deck is less than the number of forcing frequencies. Hence, PARAM, NFDOPTS, 2 can improve performance as well as reduce disk space requirement for large frequency response jobs. With PARAM, NFDOPTS, 1, frequency dependent matrices are computed at forcing frequencies.

## NHPLUS

Default = 20

In nonlinear harmonic response analysis, in order to avoid aliasing in the calculation of the Fourier coefficients, a certain number of extra evaluation points are used. NHPLUS allows the number of extra points to be defined.

## NINTPTS

Default = 10

NINTPTS requests interpolation over the NINTPTS elements closest to each grid point. NINTPTS=0 requests interpolation over all elements, which can be computationally intensive. Related parameters include BIGER, CURV, NUMOUT, OG, OUTOPT, S1G, S1M, S1AG and S1AM.

## NLAYERS

Default = 5 for CQUAD4 and CTRIAR, 7 for CQUADR and CTRIAR, 3 for CAXISYM (minimum=1, maximum=11) (SOLs 106, 129, 153, 159, and 400)

NLAYERS is used in material nonlinear analysis only and is the number of layers used to integrate through the thickness of CQUAD4, CTRIA3 CQUADR, CTRIAR and CAXISYM elements with isotropic material properties. Set NLAYERS=1 for efficiency if no bending is selected (MID2=0 or -1 on all PSHELL entries). Do not specify NLAYERS=1 if MID2 is greater than zero. A larger value of NLAYERS will give greater

accuracy at the cost of computing time and storage requirements. For CQUADR and CTRIAR, the maximum is 11.

### NLHTLS

Default = 0

See Remarks under Case Control command, [TSTRU \(Case\), 632](#).

### NLHTOL

Default = 1.0E-5

During the iteration procedure of nonlinear harmonic response, the norm of the residual load vector for the current step is divided by the norm of the residual load vector for the previously converged step; this value is then compared with NLHTOL. If the value is smaller than NLHTOL, the system is assumed to have converged.

### NLHTWK

Default = 1.0E-5

In nonlinear harmonic response analysis, if convergence is not obtained, a line search procedure is initiated to calculate a scaling factor for the displacement vector from which updated nonlinear loads are subsequently calculated. If the solution fails to converge more than 5 times in succession a new trial displacement vector is calculated using a push off factor the size of which is defined by NLHTWK. See also parameter [MXICODEO, 955](#).

### NLMAX

Default = 60

The number of suspected massless mechanisms is determined from the number of high ratio messages. If this number exceeds NLMAX, the number of trial massless mechanisms is reduced to NLMAX. This avoids an expensive debug run when there may be thousands of massless mechanisms due to systematic modeling error, such as having CONM2 elements on many grid points for which structural elements have been left out through oversight. The value of this parameter may be increased on initial debug runs where many high ratio DOFs may be present, and the user prefers to see them all at once, rather than on successive runs where only a part are removed at one time. See also [MECHFIX, 898](#).

### NLMIN

Default = 10

In the case of only one or a few high ratio DOFs more MMs may be present. More trial MM vectors are used, and those that do not indicate true problems are discarded. A smaller value could be considered on a stable model undergoing small modeling changes. See also [MECHFIX, 898](#).

### NLPACK

Default=100

NLPACK is used in the nonlinear solution sequence SOL 400 only. The SOL 400 will pack output data for NLPACK output time steps and restart data for the last time step as a single data package. For example, if NLPACK=100 (the default), then one data package have output data for 100 output time steps and restart

data for the last time step. Later usage such as restart or initial condition for later step can be performed only at data package boundaries. If NLPACK= -1, all output data for a STEP and restart data for end of the STEP are grouped into a single package. In this case, the restart can be performed only at STEP boundaries. NLPACK=0 is illegal. If NLPACK=1, each package of data on the database includes output data for one output time step and restart data. In this case, restart can be performed at every output time step. Please note that the output time step is controlled by the NO field on the Bulk Data entry, or NO field of FIXED time step control as well INTOUT field of ADAPT time step control in NLSTEP Bulk Data entry. NLPACK also influences the intermediate output selected by NLOPRM.

Please note that NLPACK is based on the number of the output load steps (concerning INTOUT in NLSTEP, and so on), but not the number of the calculated load steps. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

## NLTOL

Default =  $\begin{cases} 2 & (\text{SOL 106 and MD Nastran SOL 400}) \\ 0 & (\text{SOL 153, nonlinear heat transfer}) \end{cases}$

NLTOL establishes defaults for the CONV, EPSU, EPSP and EPSW fields of NLPARM Bulk Data entry for nonlinear static and heat transfer analysis according to the following table:

| NLTOL | Level of Accuracy  |
|-------|--------------------|
| 0     | Very high          |
| 1     | High               |
| 2     | Engineering Design |
| 3     | Preliminary Design |

See Remark 16. of the NLPARM entry for further details and corresponding default NLPARM field values.

## NMLOOP

Default = 0

In SOL 106 nonlinear static analysis, normal modes will be computed with the updated nonlinear stiffness if PARAM,NMLOOP is requested in the subcase section, it must be for all subcases. The nonlinear normal modes will be computed at the last iteration loop of each subcase in which a METHOD command appears.

## NMNLFRQ

Default = 0.0

PARAM, NMNLFRQ, Real\_Value allows users to select material or property values for frequency dependent materials or properties at a desired frequency other than the “nominal” values specified on the MATi or PBUSH, PCOMP, PCOMPG, PDAMP, or PELAS entries.

If there is only CBUSH frequency dependency and NO other type of material dependency, then Nastran will not use/update the CBUSH properties at the specified NMNLFRQ.

## NOCOMPS

Default = +1

NOCOMPS controls the computation and printout of composite element ply stresses, strains and failure indices. If NOCOMPS = 1, composite element ply stresses, strains and failure indices are printed. If NOCOMPS = 0, the same quantities plus element stresses and strains for the equivalent homogeneous element are printed. If NOCOMPS=-1, only element stresses and strains are printed. MAT1, MAT2, or MAT8 entries with MIDs>99999999 run the risk of conflicting with internally generated MAT2 entries when composites are present and should be avoided. Even if no composites are present, MIDs>99999999 signal to shell element stress computations that no STRESS is to be computed. In the case of MIDs>99999999 if the user desires stress calculations for shell elements, even if no composites are present, then PARAM, NOCOMPS, -1 is required.

Homogenous stresses are based upon a smeared representation of the laminate's properties and in general will be incorrect. Element strains are correct however.

NOCOMPS only supports conventional elements, not advanced elements. For advanced nonlinear element, only ply stress/strain will be output.

## NODCMP

Default = 0

For some type of nonlinear transient problems, including heat transfer, the decomposition of the solution matrix is not required. In order to increase efficiency, NODCMP is created to determine whether the solution matrix will be decomposed. If NODCMP=0, the solution matrix will be decomposed. If NODCMP=1, the solution matrix will NOT be decomposed. In this case, MAXBIS and DJUST on the Bulk Data entry, NLPARM, must be set zero on the Bulk Data entry. NODCMP is available for SOL 129 and 159 only.

## NOELOF

Default = -1

If NOELOF > 0, then the grid point forces (GPFORCE Case Control command) are computed along the edges of the two-dimensional elements. The default value will suppress this output.

## NOELOP

Default = -1

If NOELOP > 0, then the sum of the grid point forces (GPFORCE Case Control command) are computed parallel to the edges of adjacent elements. The effect of CBAR element offsets is not considered in the calculation of the forces. The default value will suppress this output.

## NOGPF

Default = 1

NOGPF controls the printout of grid point forces (GPFORCE Case Control command). NOGPF > 0 specifies that the grid point forces are to be printed. NOGPF < 0 suppresses the printing of the grid point forces.

## NOMSGSTR

Default = 0

If NOMSGSTR = -1, the MSGSTRES module will be skipped even though Bulk Data entries provided for it. See Part III of the *MSGMESH Analyst Guide* for a discussion of MSGSTRESS.

## NONCUP

Default = -1

In SOLs 111 and 112, NONCUP selects either a coupled or uncoupled solution algorithm in the modal dynamic response analysis. This parameter has the following meanings in both solution sequences:

- |                          |   |
|--------------------------|---|
| NONCUP = -1<br>(Default) | Use the uncoupled solution algorithm if there are no off-diagonal terms in any of the modal matrices (MHH, BHH and KHH); otherwise use the coupled solution algorithm. However, in order to ensure correct results, the coupled solution algorithm is used when large mass enforced displacement, large mass enforced velocity or SPC/SPCD enforced motion is employed in SOL 112, regardless of the existence of off-diagonal terms in the modal matrices. |
| NONCUP > -1              | Use the coupled solution algorithm unconditionally  |
| NONCUP = -2              | Use the uncoupled solution algorithm unconditionally  |

User Information Message 5222 indicates which solution algorithm is used in the analysis.

## NQSET

Default = 0

If NQSET > 0, and the PARAM entry is in Case Control, all part superelements that do not contain QSET entries, or are not referenced by SENQSET entries in the main Bulk Data Section, have NQSET generalized coordinates assigned to them. These QSET variables are functionally equivalent to those generated by SENQSET entries.

### **NUMOUT, NUMOUT1, NUMOUT2, NUMOUT3**

See S1, S1G, S1M.

### **OELMOPT**

Integer; Default=3.

PARAM,OELMOPT selects the class of elements to include in the search for grid points:

- 0 All regular elements but not PLOTEL and rigid elements.
- 1 All regular elements and PLOTEL.
- 2 All regular elements and rigid elements.
- 3 Both 1 and 2 (default).

### **OELMSET**

Integer; Default = 0

Identification number of a Case Control command SET definition. The members of the specified SET represent the identification numbers of the finite elements that are to be retained in the “reduced” op2 file element connection data block.

### **OG**

Default=0

See CURV.

### **OGEOOM**

Default = YES

See POST < 0.

### **OGRDOPT**

Integer; Default = 1

Selects the method used to create the set of grid points retained in the reduced grid point geometry data block. The default simply uses the set of grid point IDs listed in the OGRDSET Case Control SET. Set consistency is checked. OGRDOPT=2 uses the list of grid point IDs that are connected to elements in the OELMSET Case Control SET. OGRDOPT=3 merges the contents of the OGRDSET Case Control SET with the contents of the grid point list connected to the elements in the OELMSET Case Control SET. There is no consistency check for OGRDOPT=2 or OGRDOPT=3. OGRDOPT=0 turns the SET consistency check off altogether. For this case, the grid points retained are those specified in the OGRDSET SET and the elements retained are those specified in the OELMSET SET. See also PARAM,OELMOPT description.

## OGRDSET

Integer; Default = 0

Identification number of a case control command SET definition. The members of the specified SET represent the identification numbers of the grid points that are to be retained in the “reduced” op2 file grid geometry data block.

## OLDSEQ

Default = 
$$\begin{cases} -1 & \text{for non-iterative-distributed-parallel solutions} \\ 5 & \text{for iterative solutions using distributed parallel methods} \\ & (\text{NASTRAN ITER}=1 \text{ and system}(231)>0) \\ 6 & \text{if SUPER}=2 \end{cases}$$

OLDSEQ selects from the following options for resequencing:

- 1 No resequencing is performed.
- 1 Use the active/passive option.
- 2 Use the band option.
- 3 For the active/passive and the band option select the option giving the lowest RMS value of the active columns for each group of grid points.
- 4 Use the wavefront (Levy) option.
- 5 Use the Gibbs-King option even if the CPU estimate is higher than for nonsequencing.
- 6 Use the automatic nested dissection option even if the CPU estimate is higher than for no resequencing. See the following SUPER=2 description.
- 8 Semiautomatic selection. The program will compute estimates for two options that are suitable for the decomposition method selected by the PARALLEL and SPARSE keywords on the NASTRAN statement and select the option with the lowest estimate. The following table shows the suitable options for each decomposition method.

| Decomposition Method | Suitable Options |
|----------------------|------------------|
| regular              | 1 and 4          |
| parallel             | 2 and 5          |
| sparse               | 6 and 7          |

- 9 The extreme partitioning method is used to partition the model into domains.
- 10 The METIS partitioning method is used to partition the model into domains.
- 11 The MSCMLV partitioning method is used to partition the model into domains.

**Notes:**

1. The model partitioning options make sense only when running with the DOMAININSOLVER command in the Executive Control Section. For DOMAININSOLVER (PARTOPT=GRID), param,oldseq,9 is the default. For all other DOMAININSOLVER options, the default is param,oldseq,11.
2. The wavefront option does not support superelement resequencing or starting nodes. Also note that the automatic nested dissection option uses starting nodes only to establish the root of the initial connectivity tree.

If the value of OLDSEQ is changed in superelement analysis, an SEALL=ALL restart is required.

PARAM,FACTOR is used to generate the sequenced identification number (SEQID) on the SEQGP entry as follows:

$$\text{SEQID} = \text{FACTOR} * \text{GRP} + \text{SEQ}$$

where:

SEQ = generated sequence number

GRP = group sequence number

If GRP=0, use GRP(MAX)+1 where GRP(MAX) is the largest group sequence number in the database.

PARAM,MPCX controls whether the grid point connectivity created by the MPC, MPCADD, and MPCAX entries and/or the rigid element entries (e.g., RBAR) is considered during resequencing:

- 1      Do not consider the connectivity of the MPC, MPCADD, MPCAX, or rigid element entries.
- 0      Consider the connectivity of the rigid element entries only. (Default).
- >0     Consider the connectivity of the rigid element entries and the MPC, MPCADD, and MPCAX entries with the set identification number set to the value of this parameter.

PARAM,SEQOUT controls the output options as follows:

- 0      Do not generate any printed or punched output for the new sequence (Default).
- 1      Print a table of the internal/external sequence in internal order.
- 2      Write the SEQGP entries to the PUNCH file.
- 3      Perform SEQOUT=1 and 2.

PARAM,START specifies the number of the grid points at the beginning of the input sequence. The input sequence will be the sorted order of the grid point numbers including the effect of any SEQGP entries input by the user. A single SEQGP entry can be input to select the starting point for the new sequence. Otherwise, the first point of lowest connectivity will be used as the starting point.

If PARAM,SUPER<0, all grid points from the connection table that are not part of the group currently being processed are deleted. This option provides for sequencing only the interior points of a superelement. If any

superelements are present, the residual structure is not resequenced. If all of the grid points are in the residual structure, they are resequenced.

If PARAM,SUPER=0 or 1, all grid points in the connection table are considered. This option provides for the recognition of passive columns.

If PARAM,SUPER=2, then all points that are connected to multipoint constraints (via MPC entries) or rigid elements (e.g., the RBAR entry) are placed in a special group at the end of the sequence. This option also forces OLDSEQ=6 and may not be selected with other values of OLDSEQ. This option is intended primarily for models that have many active columns due to MPCs or rigid elements; e.g., a model with disjoint structures connected only by MPCs or rigid elements.

## OLDWELD

Default = NO

By default, the program uses new formulations for connector elements (CFAST, CSEAM and CWELD). To acquire identical connector results as those from previous versions before v2017.1, specify OLDWELD=YES.

## OMACHPR

Default = NO

See POST < 0.

## OMAXR

Default =  $2 \cdot \text{BUFFSIZE}$

OMAXR specifies the maximum record length of data written by the OUTPUT2 module under PARAM,POST,<0 and PARAM,OPTEXIT,-4. BUFFSIZE is a machine-dependent value defined in the *MSC Nastran Configuration and Operations Guide*. For further information, see the OMAXR parameter description under the OUTPUT2 module description in [MSC Nastran DMAP Programmer's Guide](#).

## OMID

Default = NO

To print or punch the results in the material coordinate system, set the parameter OMID to yes. Applicable to forces, strains, and stresses for TRIA3, QUAD4, TRIAR6, and QUAD8. This is available for linear analysis only. Other elements and outputs are not supported. This capability is not supported by post-processors (xdb and op2 output are not changed), by grid point stress output that assumes output is in element coordinate system, random response analysis, heat transfer analysis, stress sorting, MAXMIN Case Control, composite elements, MONPNT3 Bulk Data and external superelement data recovery in the assembly run.

## OMSGLVL

Integer; Default = 0

Set consistency check error message severity flag. The default causes FATAL messages to be generated if the grid set is not consistent with the element-related grid point set and the job is terminated. If OMSGLVL=1, the FATAL messages are reduced to WARNINGS and the job is allowed to continue.

## OPCHSET

Integer; Default = 0

SET punch request flag. If OPCHSET=1, then the list of grid points used to reduce the grid point geometry data block will be punched in case control SET definition format.

## OPGEOM

Default = -1

OPGEOM > -1 prints the aerodynamic set definitions for all degrees-of-freedom. To print structural degree-of-freedom sets, please see [USETPRT](#).

## OP2GM34

Default = TRUE

By default, the GEOM3 and GEOM4 are written in OP2 file. To avoid GEOM3 and GEOM4 output, set PARAM, OP2GM34, FALSE.

## OPGTKG

Default = -1

OPGTKG > -1 prints the matrix for the interpolation between the structural and aerodynamic degrees-of-freedom.

## OPPHIB

Default = -1

In the flutter (SOLs 145 and 200) and aeroelastic (SOLs 146 and 200) solution sequences, OPPHIB > -1 and a DISPLACEMENT request in the Case Control Section will output the real vibration modes with the structural displacement components transformed to the basic coordinate system.

## OPPHIPA

Default = -1

In the flutter (SOLs 145 and 200) and the dynamic aeroelastic (SOL 146) solution sequences, OPPHIPA > -1 and a DISPLACEMENT command in the Case Control Section will output the real vibration modes at all degrees-of-freedom, including the aerodynamic degrees-of-freedom in the global coordinate system. Use PARAM,OPPHIPB to output in the basic system.

## OPTEXIT

Default = 0

In SOL 200, especially during the checkout of the analysis model and the design optimization input data (design model), it may be desirable to exit the solution sequence at certain points before proceeding with full optimization. OPTEXIT may be set to values of 1 through 7 and -4. The DSAPRT Case Control command overrides the specification of PARAM,OPTEXIT,4, -4, or 7. The description of OPTEXIT values follow.

| OPTEXIT<br>Value | Description  |
|------------------|--|
| 0                | Do not exit. Proceed with optimization.  |
| 1                | Exit after the initialization of the analysis and design model but before finite element analysis begins.  |
| 2                | Exit after finite element analysis and initial design response and shape basis vector processing.  |
| 3                | Exit after design constraint evaluation and screening.   |
| 4                | Exit after design sensitivity analysis and print the matrix of design sensitivity coefficients (DSCM2). This is equivalent to the DSAPRT (UNFORM,END=SENS) Case Control command.   |
| -4               | Exit after design sensitivity analysis and write the data blocks related to sensitivity coefficients (DSCM2 and DSCMCOL) to an external file using the OUTPUT2 and OUTPUT4 modules. This is equivalent to the DSAPRT (NOPRINT,EXPORT END=SENS) Case Control command. See related parameters ITAPE, IUNIT, and OMAXR.   |
| 5                | Exit after the first approximate optimization of the design model.   |
| 6                | Exit after the first update of the analysis model based on the first approximate optimization of the design model.   |
| 7                | Compute and output design sensitivity coefficients at the end of normal program termination: hard convergence, soft convergence, or maximum design cycles. This is equivalent to the DSAPRT (UNFORM,START=LAST) Case Control command. If the final design is a discrete design optimization, no sensitivity is performed and the OPTEXIT=7 request is not honored. |

## OPTION

Default = ABS

See SCRSPEC.

## OSETELE

Default = 2

See [AUTOGOUT, 793](#).

## OSETGRD

Default = 1

See [AUTOGOUT, 793](#).

## OSWELM

Default = -1

Offset for identification numbers of internally generated RBE3 rigid body elements (generated by all formats of CFAST and CSEAM as well as CWELD with formats ELEMID, GRIDID, ELPAT and PARTPAT). By default, OSWELM=-1, the numbering starts with SYSTEM(182) + 1. The default of system cell 182 is SYSTEM(182)=100,001,001. If the user defines OSWELM > 0, then the numbering starts with OSWELM + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD element, a pair of RWELD constraint elements is generated if MSET=ON is specified, see the entry PWELD, 3329 for an explanation.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated RBE3 rigid body elements (generated by CWELD and CFAST, all formats) and CONM2 mass elements (generated by CFAST when it has nonzero mass). The default behavior in SOL 400 is the same as in the linear solution sequences.

## OSWPPT

Integer, Default = -1

Offset for internally generated grid identification numbers for connector elements (GA and GB for CWELD and CFAST if these are not specified and all auxiliary grids for CWELD, CFAST and CSEAM). By default, OSWPPT=-1, the numbering starts with SYSTEM(178) + 1. The default of system cell 178 is SYSTEM(178)=101,000,000. If the user provides OSWPPT > 0, then the numbering starts with OSWPPT + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD or CFAST element, a pair of grid points GA and GB is generated internally if the formats ELEMID, GRIDID, ELPAT or PARTPAT are used and if no identification numbers for GA and GB are specified, see the entry CWELD, 1696 and CFAST, 1505 for a definition of the formats.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated grids of connector elements CWELD and CFAST (GA and GB if these are not specified in the bulk data input and all auxiliary grids). The default behavior in SOL 400 is the same as in the linear solution sequences.

## OUGCORD

See POST.

## OUGSPLIT

Default=YES

By default, displacements, velocities, and accelerations are written to separate tables.

PARAM,OUGSPLIT,NO will combine these tables into a single table and is intended for third-party post-processors which are not able to read the separate tables. The value of OUGSPLIT does not affect the results in the f06 file.

## OUNIT1

Default = Value of OUNIT2

For PARAM,POST,-1 and -2 defines the unit that geometry data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

## OUNIT2

Default = 12

For PARAM,POST,-1 and -2 defines the unit that results data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

## OUTOPT

Default = 0

See CURV.

## PACINFID

Integer, no default.

This parameter is used only in data recovery operations for elements in acoustic field point mesh bulk data. PACINFID is the ID of the PACINF entry that contains the location of the pole used to determine element orientation. Data recovery operations require that the element surface normal vector point away from the pole location. This parameter is required only if the acoustic model references multiple PACINF bulk data entries and the pole locations on the PACINF entries are not coincidental.

## PANELMP

Replaced by a keyword on the FLSPOUT Case Control command.

## PATPLUS

Default = NO

PARAM,PATPLUS,YES may be used with PARAM,POST,0 to allow the user to also write data blocks GPFORCE, ESE, EKE, EDE, WETSENS, and ESE-(with PARAM,XFLAG,2) to a Fortran unit as is done under PARAM,POST,-1.

## PATVER

Default = 3.0

See POST = -1.

## PCOMPRM

Default = 0

PCOMPRM controls the computation and printout of composite element extreme ply stresses, strains, failure Indices and strength ratios. Note that extreme ply output encompasses ply stresses, ply strains, ply failure indices and ply strength ratios together. Punch output is not available for extreme ply results.

PCOMPRM = 0 (default) , STANDARD composite ply responses are output.

PCOMPRM = 1 STANDARD & EXTREME composite ply responses are output  
 PCOMPRM = 2 ONLY EXTREME composite ply responses are output

## PDRMSG

Default = 1

PDRMSG controls the printout of messages associated with deformed plots, including error messages.  
 PDRMSG = 0 suppresses the printout. Contour values will not be displayed unless the default value is used.

## PEDGE

Default = 0

Cubic edges of p-elements can be created with the FEEDGE Bulk Data entry by defining two vertex grids and two points in between. By default, the two points on an edge are moved to the parametric 1/3 and 2/3 locations of the edge. For PEDGE = 1 the points are not moved. MSC Patran V7 generates points so that adjacent edges are C1 continuous. These points should not be moved. Therefore, MSC Patran generates a Bulk Data Section with PARAM,PEDGE,1 if p-elements are in the model.

## PEMFRIM

PARAM,PEMFRIM,1 is used to request full Reduced Impedance Matrix (RIM) from Actran, instead of lower-triangular RIM. By default (PARAM,PEMFRIM,0), Actran returns lower-triangular RIM. Note that PEMFRIM will increase disk space demand of PEM job. In addition, OOC field of ACPEMCP will cause Actran to generate full RIM when set to an integer greater than 1.

## PEMMAPC

PEMMAPC is used to generate lists of GRD IDs of the coupling surfaces. 'PARAM,PEMMAPC,1' (default is 0) will produce following files in 'mapping\_control' subdirectory of job submit directory.

| For   | SET ID   | Generic File name   |
|---|--|---|
| Structure GID coupled with all TRMCs        | 100000   | <deckname>.<job seq>.x_nas.structure_component1.set                               |
| Fluid GID coupled with all TRMCs            | 200000   | <deckname>.<job seq>.x_nas.fluid_component2.set                                   |
| Structure GID coupled with TRIMID=xxxx only | y00018xxxx for GLUED or z00018xxxx for SLIDING | <deckname>.<job seq>.x_nas.structure_component1.trim_coupling.<xxxx>.<y or z>.set |
| Fluid GID coupled with TRIMID=xxxx only     | Y00028xxxx for OPEN or Z00028xxxx for IMPER    | <deckname>.<job seq>.x_nas.fluid_component2.trim_coupling.<xxxx>.<Y or Z>.set     |
| TRIMID=xxxx GID coupled with structure      | xxxx90001                                      | <deckname>.<job seq>.x_nas.interface<xxxx>.surface.1.set                          |
| TRIMID=xxxx GID coupled with fluid          | xxxx90002                                      | <deckname>.<job seq>.x_nas.interface<xxxx>.surface.2.set                          |

Guidelines for PARAM,PEMMAPC,1

1. PARAM,PEMMAPC,1' must be placed in the main bulk data section or under 'BEGIN BULK' to be effective.
2. Without 'PARAM,PEMNPART,n', PEMMAPC logic has a TRIM ID limit of less than 10000.
3. With 'PARAM,PEMNPART,n', PEMMAPC login has a TRIM ID limit of less than 100.

## PEMNCOP4

PARAM,PEMNCOP4,1 (default) is used to request each Reduced Impedance Matrix (RIM) from Actran to be written into individual file. With PARAM,PEMNCOP4,0, Actran writes all RIMs into a single file. Note that PARAM,PEMNCOP4,1 can reduce footprint on disk space of large PEM jobs.

## PEMNPART

PEMNPART can be utilized to partition a large trim component. The default value for PEMNPART is 1. 'PARAM,PEMNPART,n' will partition a large trim component into 'n' parts. The guidelines are:

1. PARAM,PEMNPART,n' must be placed under 'BEGIN TRMC=xx' to be effective for trim component 'xx'.
2. Each trim component can have its own 'PARAM,PEMNPART,n'
3. When value of 'n' for PEMNPART is greater than 1, new trim components are generated with ID equal to 'xx\*100+<seq no> where <seq no> goes from 0 to n-1.
4. For no loss of accuracy, setting 'n' to a value equal to the number of disconnected parts of a large trim component.
5. For accepting loss of accuracy to run on a machine with limited memory, setting 'n' to a value larger than the number of disconnected parts of a large trim component.
6. 'PARAM,PEMNPART,n' with n>1 should be utilized only as an alternative for running PEM jobs with large trim component and insufficient memory on the computer. Other alternatives for running PEM job with large trim component and insufficient memory are:
  - a. Split large trim component into smaller trim components and/or
  - b. Acquire more memory for the computer
7. For a trim component with 'PARAM,PEMNPART,n' where n>1 and 'PARAM,TRMBIM,physical', data recovery will be skipped for all partitioned trim components.

## PEMSKIN

'PARAM,PEMSKIN,1' is used for Actran to couple the trim bodies only to the surface of the cavity and will not search any node inside the cavities. The default value for PEMSKIN is 0 which allows Actran to target both surface and interior nodes of the cavities for trim bodies coupling.

## PENFN

Default = 1.0e+5 (See LMFAC)

## PERCENT

Default = 40

See CQC under [SCRSPEC, 987](#).

## PH2OUT

Default = 0, SOL 400 only

For nonlinear solution sequence, SOL 400, in addition to the regular phase III output, the user can also request the phase II output. This is useful when the run is terminated abnormally before the phase III outputs are formatted and printed. The phase II output consists of all outputs requested by the Case Control commands in the input file and prints in sort1 format. If there's no PH2OUT, MSC Nastran outputs phase III outputs only. This is the regular output. If PH2OUT=1, MSC Nastran outputs phase II outputs only. In this case, there will be no output for the upstream superelements. If PH2OUT=3, MSC Nastran outputs both phase II and phase III outputs. In this case, some of the outputs for the residual structure may be redundant.

Case Control command, NLOPRM OUTCTRL, takes precedence over PH2OUT. In other words, when NLOPRM OUTCTRL is present, PH2OUT is no longer functional. NLOPRM OUTCTRL=SOLUTION is equivalent to PH2OUT=1 and NLOPRM OUTCTRL=(STD,SOLUTION) to PH2OUT=3.

## PKRSP

Default = -1

If PKRSP=0, the magnitude of the output quantities at the time of peak acceleration of the modal variables is output. This option is available only for modal transient analysis.

## PLTMSG

Default = 1

PARAM,PLTMSG,0 suppresses messages associated with undeformed plot requests, including error messages.

## POST

Default = 9999999

If PARAM,POST,0, then the following parameters and discussion apply (not support in SOL 400):

The data blocks often used for pre- and postprocessing will be stored in the database and also converted, by the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)), to a format suitable for processing by MSC Patran. These data blocks include input data related to geometry, connectivity, element and material properties, and static loads; they also include output data requested through the Case Control commands OLOAD, SPCF, DISP, VELO, ACCE, THERMAL, ELSTRESS, ELFORCE, FLUX, GPSTRESS, GPFORCE, ESE, GPSDCON, and ELSDCON.

The converted data is written to logical FORTRAN units, which may be assigned to physical files in the File Management Section. The FORTRAN unit numbers are specified by the parameters GEOMU, POSTU, and LOADU. By default, all data is written to the logical FORTRAN unit indicated by GEOMU. If LOADU > 0, static load data may be diverted to another unit indicated by LOADU. If POSTU > 0, then output data requested with the Case Control commands listed above will be diverted to the logical unit indicated by POSTU. See [Database Concepts in the MSC Nastran Reference Guide](#) for the procedure for assigning physical files.

By default, if converted data already exists on the files indicated by GEOMU, POSTU, and LOADU, then the DBC module will overwrite the old data. If this is not desirable, then PARAM,DBCOVWRT,NO must be entered. The parameters MODEL and SOLID may be used to store more than one model and solution in the graphics database. These parameters are not supported by MSC Patran.

PARAM,DBCDIAG > 0 requests the printing of various diagnostic messages from the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)) during the data conversion. By default, no messages are printed.

If PARAM,PATPLUS,YES is specified along with PARAM,POST,0 then the PARAM,POST,-1, operation will also be performed.

If PARAM,POST,< 0 or 1, then the following parameters and discussion apply:

- PARAM,POST,1 outputs the appropriate files for the SimXpert and Patran programs and should be used to obtain results from the latest result data blocks from SOL 400 and other recent MSC Nastran developments such as NEF.
- PARAM,POST,-1 outputs the appropriate files for the Patran program (PARAM,POST,1 should be used for more recent versions of MSC Nastran > 2012).
- PARAM,POST,-2 outputs the appropriate files for the Siemens I-deas® program.
- PARAM,POST,-4 outputs the files indicated below along with OPHIG for the MSC\_NF interface by LMS International.
- PARAM,POST,-5 outputs the files indicated in the table below along with LAMA and OPHG1 for the FEMtools interface by Dynamic Design Solutions.
- POST=-4 and -5 are intended for SOL 103 only.
- PARAM,POST,-6 outputs the files indicated below for Siemens Unigraphics®.
- PARAM,POST,-7 outputs the files for Free Field Technologies.

An OUTPUT2 file for FORTRAN unit 12 in binary format is automatically created in the same directory and with the same name as the input file and with the extension ".op2". For example, if the input file is fender.dat then the OUTPUT2 file will be called fender.op2.

An ASSIGN statement is required in the FMS Section only if neutral file format is desired as follows:

ASSIGN OP2='filename of FORTRAN file' FORM

Geometry data blocks are output with PARAM,OGEOM,YES (Default) and are written to a FORTRAN unit specified by PARAM,OUNIT1 (Default = OUNIT2) for POST = -1, -2, -4, and -6.

PARAM,OUNIT2K (Default = 91) specifies the unit number for KELM and KDICT with PARAM,POST,-5. PARAM,OUNIT2M (Default = 92) specifies the unit number for MELM and MDICT with PARAM,POST,-5. Note that PARAM,POST,-5 is not supported with DMP. See the following table for the specific geometry data blocks written for different values for POST.

See also the PARAM,POSTEXT description for additional data blocks written to the .op2 file.

| POST       |     |     |     |     | Geometry Data Block | Description                           |
|------------|-----|-----|-----|-----|---------------------|---------------------------------------|
| -1,1 or -7 | -2  | -4  | -5  | -6  |                     |                                       |
| YES        | NO  | NO  | NO  | NO  | GEOM1S,<br>GEOM1VU  | Grid Point Definitions (Superelement) |
| NO         | YES | YES | NO  | YES | CSTM                | Coordinate System Transformations     |
| NO         | YES | YES | NO  | YES | GPL                 | Grid Point List                       |
| NO         | YES | YES | NO  | YES | GPDT                | Grid Point Definitions                |
| NO         | YES | YES | NO  | NO  | EPT                 | Element Properties                    |
| NO         | YES | YES | NO  | NO  | MPT                 | Material Properties                   |
| YES        | YES | YES | NO  | NO  | GEOM2               | Element Definitions                   |
| YES*       | YES | NO  | NO  | NO  | GEOM3               | Load Definitions                      |
| YES*       | YES | NO  | NO  | NO  | GEOM4               | Constraint Definitions                |
| YES        | NO  | NO  | NO  | NO  | DIT                 | Dynamic Table Input                   |
| YES        | NO  | NO  | NO  | NO  | DYNAMICS            | Dynamic Loads Definition              |
| NO         | NO  | YES | YES | NO  | KDICT               | Element Stiffness Dictionary          |
| NO         | NO  | YES | YES | NO  | KELM                | Element Stiffness Matrices            |
| NO         | NO  | YES | YES | NO  | MDICT               | Element Mass Dictionary               |
| NO         | NO  | YES | YES | NO  | MELM                | Element Mass Matrices                 |
| NO         | NO  | NO  | NO  | YES | ECTS                | Element Connections                   |
| YES        | NO  | NO  | NO  | NO  | VIEWTB              | View Element Table                    |
| YES        | NO  | NO  | NO  | NO  | EDOM                | Design Model Input                    |
| YES        | NO  | NO  | NO  | NO  | GEOM2S,<br>GEOM2VU  | Same as GEOM2 for superelements       |
| YES        | NO  | NO  | NO  | NO  | CSTMS               | Same as CSTM for superelements        |
| YES        | NO  | NO  | NO  | NO  | EPTS                | Same as EPT for superelements         |
| YES        | NO  | NO  | NO  | NO  | MPTS                | Same as MPT for superelements         |

\* To avoid having GEOM3 and GEOM4 written if POST=-1, set PARAM,OP2GM34,FALSE.

PARAM,OMACHPR,NO selects the single precision format for GPDT, CSTM, and GEOM1.

PARAM,OMACHPR,YES selects the machine precision format. With PARAM,POST,<0,  
PARAM,OMACHPR,NO is the default. With PARAM,POST,1, PARAM,OMACHPR,YES is the default.

For PARAM,POST = -1 and -2, results data blocks are output to a FORTRAN unit specified by  
PARAM,OUNIT2 (Default = 12). This parameter is allowed to vary between superelements. In buckling  
solution sequence (SOL 105), a unique value of OUNIT2 should be specified for the buckling subcase. For  
all other solution sequences or within a given analysis type, changes to PARAM,OUNIT2 between subcases  
is not supported. See the POST Case Control command to change the unit number between subcases. See  
also the related parameter OMAXR.

By default the displacements are output in the global coordinate system. To output in the basic coordinate system, specify PARAM,OUGCORD,BASIC.

#### PARAM,POST,-1 or 1: Results Data Blocks for Patran and SimXpert

By default, the following data blocks are output under PARAM,POST,-1. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file. It is recommended when processing large post files in Patran and SimXpert that the following environment variable setting is used when starting the session: DRANAS\_NAST\_MEM=2048MB.

| PARAM,PATVER |       | Parameter Name | Case Control                                  | Data Block Name                           | Description   |
|--------------|-------|----------------|---|---|---|
| < 3.0        | ≥ 3.0 |                |   |   |   |
| YES          | YES   | OQG            | SPCFORCE                                      | OQG1                                      | Forces of single-point constraint   |
| YES          | NO    | OUG            | DISP  | OUGV1PAT                                  | Displacements in the basic coordinate system  |
| YES          | YES   | OUG            | DISP  | OUGV1                                     | Displacements in the global coordinate system   |
| YES          | NO    | OES            | STRESS  | OES1                                      | Element stresses (linear elements only)   |
| YES          | NO    | OEF            | FORCE   | OEF1                                      | Element forces or heat flux (linear elements only)  |
| YES          | YES   | OEE            | STRAIN  | OSTR1                                     | Element strains   |
| YES          | YES   | OGPS           | GPSTRESS                                      | OGS1                                      | Grid point stresses   |
| YES          | YES   | OESE           | ESE   | ONRGY1                                    | Element strain energy   |
| YES          | YES   | OGPF           | GPFORCE                                       | OGPFB1                                    | Grid point force balance table  |
| NO           | YES   | OEFX           | FORCE   | OEF1X                                     | Element forces with intermediate (CBAR and CBEAM) station forces and forces on nonlinear elements       |
| NO           | YES   | OESX           | STRESS  | OES1X                                     | Element stresses with intermediate (CBAR and CBEAM) station stresses and stresses on nonlinear elements |
| NO           | YES   | OPG            | OLOAD   | OPG1                                      | Applied static loads  |
| NO           | YES   | OCMP           | STRESS  | OES1C                                     | Ply stresses  |
| NO           | YES   | OCMP           | STRAIN  | OSTR1C                                    | Ply strains   |
| NO           | YES   | none           | DISP<br>SPCFORCE<br>FORCE<br>STRESS<br>STRAIN | OUPV1<br>OQP1<br>DOEF1<br>DOES1<br>DOSTR1 | Scaled Response Spectra   |
|              |       |                | none  | LAMA                                      | Nonlinear Buckling  |
| NO           | YES   | none           | DISP<br>OLOAD                                 | OCRUG<br>OCRPG                            |   |
| NO           | YES   | none           | NLSTRESS                                      | OESNLXR                                   | Nonlinear static stresses   |

| PARAM,PATVER |       | Parameter Name | Case Control | Data Block Name | Description                   |
|--------------|-------|----------------|--------------|-----------------|-------------------------------|
| < 3.0        | ≥ 3.0 |                |              |                 |                               |
| NO           | YES   | none           | BOUTPUT      | OESNLBR         | Slideline stresses            |
| NO           | YES   | none           | NLLOAD       | OPNL1           | Nonlinear loads               |
| NO           | YES   | none           | STRESS       | OESNLXD         | Nonlinear transient stresses  |
| NO           | YES   | none           | none         | ERRORN          | p-element error summary table |

**PARAM,POST,-2: Results Data Blocks for Siemens I-deas®**

By default, the following data blocks are output under PARAM,POST,-2. By default, the displacements are output in the basic coordinate system. To output in the global coordinate system, specify PARAM,OUGCORD,GLOBAL. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file.

| PARAMeter Name | Case Control | Results Data Block Name | Description                                   |
|----------------|--------------|-------------------------|---|
| OQG            | SPCFORCE     | OQG1                    | Forces of single-point constraint             |
| OUG            | DISPLACE     | BOUGV1                  | Displacements in the basic coordinate system  |
|                |              | BOPHIG                  | Eigenvectors in the basic coordinate system   |
|                |              | OUGV1                   | Displacements in the global coordinate system |
|                |              | TOUGV1                  | Grid point temperatures                       |
| OES            | STRESS       | OES1                    | Element stresses (linear elements only)       |
| OEF            | FORCE        | OEF1                    | Element forces (linear elements only)         |
|                | FLUX         | HOEF1                   | Element heat flux                             |
| OEE            | STRAIN       | OSTR1                   | Element strains                               |
| OESE           | ESE          | ONRGY1                  | Element strain energy                         |
| OCMP           | STRESS       | OEFIT                   | Failure indices                               |
|                | STRESS       | OES1C                   | Ply stresses                                  |
|                | STRAIN       | OSTR1C                  | Ply strains                                   |
| OUMU           | ESE          | LAMA                    | Eigenvalue summary                            |
|                |              | ONRGY2                  | Element strain energy                         |
| OEFX           | FORCE        | OEF1X                   | Element forces (nonlinear elements only)      |
| OESX           | STRESS       | OES1X                   | Element stresses (nonlinear elements only)    |
| none           | none         | ODELBGPD                | Shape optimization geometry changes           |

[PARAM, POST, -4: Results Data blocks for LMS International/MSC\\_NF](#)

By default, the following data blocks are output under PARAM,POST,-4. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file.

| PARAMeter Name | Case Control | Data Block Name | Description                                   |
|----------------|--------------|-----------------|---|
| OUG            | DISPLAC      | OPHIG           | Eigenvectors in the global coordinate system. |

[PARAM, POST, -5: Results Data blocks for Dynamic Design Solutions/FEMtools](#)

By default, the following data blocks are output under PARAM,POST,-5. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file. PARAM,OUNIT2O (Default51) specifies the unit number of the OUTPUT2 file.

| PARAMeter name | Case Control | Data Block Name | Descriptions                                  |
|----------------|--------------|-----------------|---|
| OUG            | DISPLAC      | OUGV1           | Eigenvectors in the global coordinate system. |
|                |              | LAMA            | Eigenvalue summary.                           |

[PARAM, POST, -6: Results Data Blocks for Siemens Unigraphics®](#)

By default, the following data blocks are output under PARAM,POST,-6. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that displacements not be written to the OUTPUT2 file.

| PARAMeter Name | Case Control | Data Block Name | Description  |
|----------------|--------------|-----------------|--|
| OQG            | SPCFORCE     | OQG1            | Forces of single-point constraints   |
| OUG            | DISPLAC      | OUGV1           | Displacements  |
| OES            | STRESS       | OES1            | Element stresses   |
| OEF            | STRESS       | OEF1            | Element forces   |
| OEE            | STRAIN       | OSTR1           | Element strains  |
| OESX           | STRESS       | OES1X           | Element stresses with intermediate station stresses and stresses on nonlinear elements |
| OEFX           | STRESS       | OEF1X           | Element forces with intermediate station forces and forces on nonlinear elements       |
| OPG            | OLOAD        | OPG1            | Applied static loads   |
| none           | none         | DVPTAB          | Designed property table  |

| PARAMeter Name | Case Control | Data Block Name | Description               |
|----------------|--------------|-----------------|---------------------------|
| none           | none         | OPTPRMG         | Optimization parameters   |
| none           | none         | PROPO           | Final element properties  |
| none           | none         | DBCOPT          | Optimization summary data |

## PARAM,POST,-7

PARAM,POST,-7 Results Data Blocks for Free Field Technologies.

Except eigenvectors, all other output data blocks are the same as PARAM,POST,-1. For eigenvectors, the matrix form is written to the OUTPUT2 file.

## POSTEXT

Default = NO

Under PARAM,POST,-1 and -2, and if PARAM,POSTEXT,YES is specified, then the following data blocks are also written to the .op2 file specified by PARAM,OUNIT2.

| Data Block Name | PARAM POST | Description   |
|-----------------|------------|---|
| FRL             | -1, -2     | Frequency response list (modal frequency response only).  |
| BHH             | -1, -2     | Modal damping matrix (modal frequency response only).     |
| KHH             | -1, -2     | Modal stiffness matrix (modal frequency response only).   |
| BGPDT           | -1         | Basic grid point definition table.                        |
| PVT0            | -1, -2     | User parameter value table                                |
| CASECC          | -1         | Case Control table  |
| EQEXIN(S)       | -1         | Equivalence external to internal grid ID table            |
| CLAMA           | -1, -2     | Complex eigenvalue table                                  |
| OEDE1           | -1, -2     | Element energy loss output table                          |
| OEKE1           | -1, -2     | Element kinetic energy output table                       |
| OUGV2           | -1, -2     | Displacement output table in SORT2                        |
| PSDF            | -1, -2     | Power spectral density table                              |
| OGPWG           | -1, -2     | Grid point weight generator output table                  |
| TOL             | -1, -2     | Time output list  |
| OPHSA           | -1, -2     | Solution set eigenvectors (modal frequency response only) |
| LAMA            | -1         | Eigenvalue summary table                                  |
| ONRGY2          | -1         | Element kinetic energy (obsolete)                         |
| PSDFH           | -1, -2     | Power spectral density table for modal coordinates        |
| DSCM2           | -1, -2     | Design sensitivity coefficient matrix                     |
| DSCMCOL         | -1, -2     | Design sensitivity parameters table                       |

## POSTADF

Default = NO

This parameter is used to export the results of frequency analysis responses and participation factors by PFMODE, PFPANEL, and PFGRID in ADF format file. This exportation is requested by setting this parameter to YES. The unit information of the exported functions should be defined by DTI, UNITS statement for this function (see ADAMSMNF statement). If not defined, SI unit will be assumed with the warning message. The ADF file is defined as input data name with the extension “afu” automatically. Alternatively the name can be defined with ASSIGN ADFFILE= statement in FMS section. For the participation factor output, FILTER=0. option should be applied in PFMODE and PFPANEL.

## POSTU

Default = -1

See POST=0.

## PREFDB

Default = 1.0

See ACOUT.

## PRGPST

Default = NO

PRGPST is replaced by the PRINT keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,PRGPST.

## PRINT

Default = YES

PARAM,PRINT,NO suppresses the automatic printing of the flutter summary in flutter analysis.

## PROUT

Default = -1

PARAM,PROUT,-1 suppresses execution and printout from the ELTPRT module. PARAM,PROUT,-1 prints a list of all elements sorted on EID and summary tables giving the range of element identification numbers for each element type.

ELTPRT will provide an information message when it detects any duplicate element ID. For the case of CONV and CHYBDi elements, duplicate element IDs are by design. The information message provided by ELTPRT can be safely ignored for these elements.

## PRPA

Default = 1.0E37

## PRPJ

PRPA and PRPJ control the printout of intermediate load matrices for diagnostic purposes during superelement assembly. If the value of PRPA (or PRPJ) is positive, all terms larger in magnitude than the

value are printed. If the value of PRPA (or PRPJ) is negative, all terms smaller in magnitude than the value are printed. The default value requests no printout. PARAM,IRES,1 must be present for these parameters to be effective. The PA matrix contains the internal loads transmitted to the downstream superelement. The PJ matrix contains external loads applied on the superelement; that is, it has the same content as the data produced by the Case Control command OLOAD. All of this data may be obtained on restart using the SELR Case Control command option. A related parameter is IRES.

## PRPHIVZ

Default = 1.0E37

PRPHIVZ controls the printout of the PHIVZ matrix that contains the component mode eigenvectors of the model. It includes all degrees-of-freedom with motion except the m-set, eliminated for multipoint constraints. The FAPPROX matrix contains the square root of the diagonal terms of the generalized stiffness matrix divided by  $2\pi$ . For fixed-boundary solutions, it is a good approximation for the natural frequencies of the component. For free- or mixed-boundary solutions, it is of a lower value than the natural frequencies. All terms larger than PRPHIVZ in both matrices will be printed (i.e., PARAM,PRPHIVZ,0.0 causes all terms to be printed).

## PRTMAXIM

Default = NO

PRTMAXIM controls the printout of the maximums of applied loads, single-point forces of constraint, multipoint forces of constraint, and displacements. The printouts are titled “MAXIMUM APPLIED LOADS”, “MAXIMUM SPCFORCES”, “MAXIMUM MPCFORCES”, and “MAXIMUM DISPLACEMENTS”.

## PRTRESLT

Default = YES

PRTRESLT controls the printout of the resultants of applied loads, single-point forces of constraint, and multipoint forces of constraint. The printouts are titled “OLOAD RESULTANTS”, “SPCFORCE RESULTANTS”, and “MPCFORCE RESULTANTS”.

## PSENPCH

Default = NO, for SOL 200 job with PART SEs only.

Setting PSENPCH to YES causes updated Bulk Data entries of a PART SE for a design cycle punched to a separate file named as follows

JOBNAME\_psex<sub>xx</sub>\_yy.pch – where xx is the PART SE ID and yy is the design cycle.

## PVALINIT

Default = 1

Starting p-value in a p-adaptive analysis restart.

## Q

Default = 0.0

Q specifies the dynamic pressure. Q must be specified in aeroelastic response analysis (SOLs 146), and the default value will cause a User Fatal Message.

## RADMOD

Default = YES

This parameter only applies to heat transfer solution sequences for SOLs 153, 159 and 400. The parameter, RADMOD, only affects radiation heat transfer problems. The radiation matrix is modified to avoid the temperature overshoot with a coarse mesh. If the user wants the program to skip this operation (modify the radiation matrix) then insert PARAM,RADMOD,NO.

## RBSETPRT

Default=0

This parameter controls the printout of rigid element IDs that are included in the analysis. Setting PARAM,RBSETPRT to a value not equal to zero activates the printing of rigid element IDs. Note that this function works in conjunction with the rigid element selection via MPC=n in case control and SET3,n,RBxx in bulk data.

## RESLOPT

Default = 8

RESLOPT's default value provides component-level force summary output for model checkout (PARAM, CHECKOUT, YES), loads generation (OLOAD output), and SPC and MPC forces. Setting RESLOPT to a value of 1 produces abbreviated output formats only.

## RESVEC

This parameter and the related parameters RESVINER, RESVSO, RESVSE, and RESVSLI are obsolete or replaced by options on the RESVEC Case Control command.

## RFORSET

Default = 0

A PARAM, RFORSET, n where n is a Bulk Data SET1 entry, allows the user to apply rotational force loading (RFORCE) to just a portion of the structure. RFORSET like RFORCE loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the rotational force loading is to be applied. RFORSET can appear in either the **Bulk Data** or **ABOVE** or **IN the 1st Subcase**. A RFORSET appearing in any other Subcase will be ignored and may cause incorrect results. A RFORSET appearing in or above the first subcase takes precedence over a RFORSET appearing in Bulk Data. See also the companion GRAVSET entry. If there is no RFORCE loading called out, then PARAM, RFORSET, n needs to be removed from the run.

## RHOCP

Default = 1.0

This parameter represents a scale factor used in the computation of ERP in units of dB. See the ERPPREFDB parameter. This parameter may alternatively be set using the ERP Case Control command.



## RIMINTP

RIMINTP can be used to select the interpolation method for Reduced Impedance Matrix(RIM) for PEM and ACTRIM. The input options for RIMINTP are LINEAR and LOG10. The default is LINEAR.

## RKSHEME

Default = Remark 1., SOL 700 only

Defines the type of time-integration scheme used in the Riemann solution-based Euler solvers.

### Format:

PARAM,RKSHEME,NUMBER

### Example:

PARAM,RKSHEME,3

NUMBER      Number of Runge-Kutta stages. (Integer > 0)

### Remarks:

1. This parameter can be used in combination with the original Roe solver and the improved full 2<sup>nd</sup> order fluid- and gas Euler solver. The default number of stages depends on the spatial accuracy of the solution scheme. One stage is used for first order spatial accuracy, and three stages for second order spatial accuracy.
2. It is recommended that the user leave the setting to the default values as defined depending on the selected spatial accuracy of the solution.
3. It has been found that in some cases with fluid flows, it may be necessary to manually limit the time step to a fixed and lower value than the scheme determines. These occasions are noticeable when you view a contour plot of the pressure. The user may find that the pressure field looks like a checkerboard. This is then caused by a numerical instability due to a time step that is too large for the fluid flow. The face fluxes are always correct but the element values get decoupled due to the time integration instability. Lowering the time step to about half the maximum value the scheme determines solves this problem. Please note that this significantly slows down the computation! Since this problem rarely occurs we have chosen not to automatically limit the time step to a lower value for performance reasons. However, the user needs to be aware that the analysis may exhibit the aforementioned behavior.

## RMSINT

Default = LINEAR for the trapezoidal approximation.

RMSINT specifies the interpolation method for numerical integration when computing both RMS (Root Mean Square) and N0 (Number of Zero Crossings or Mean Frequency) from PSDF (Power Spectral Density Function). RMSINT = LINEAR requests the trapezoidal approximation, which is the existing Nastran approach. RMSINT = LOGLOG requests the Log-Log interpolation.

## ROHYDRO

Default = Remark 3., SOL 700 only

Defines the minimum density for hydrodynamic, single-material Eulerian elements.

### Format:

PARAM,ROHYDRO,VALUE

### Example:

PARAM,ROHYDRO,1.E-6

VALUE            Density cutoff. (Real > 0.0)

### Remarks:

1. Hydrodynamic, single-material Eulerian elements with a density less than ROHYDRO are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than ROHYDRO, then no transport is done.
  - b. If the density of element A after transport is less than ROHYDRO, then all of the mass is transported to element B.
3. By default, the cutoff density for hydrodynamic Eulerian elements is set to 1.E-5 times the material reference density. For the Riemann solution-based solvers, the default is set to 1.E-6 times the reference density.
4. Please note that this parameter has a different effect in the Riemann-solution based Euler solvers. The elements are never viewed as empty, but a small amount of mass (equal to the element's volume times the cut-off density value) remains in the element. All other state variables (velocity, energy and pressure are reset to zero for these types of elements). For fluid flows where you wish to model cavitation, use Tait's equation of state with a critical density at which the pressure remains constant and the fluid cavitates.

## ROMULTI

Default = Remark 3., SOL 700 only

Defines the minimum density for multimaterial Eulerian elements.

### Format:

PARAM,ROMULTI,VALUE

### Example:

PARAM,ROMULTI,1.E-6

VALUE      Density cutoff. (Real &gt; 0.0)

**Remarks:**

1. Multimaterial Eulerian elements with a density less than ROMULTI are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of a specific material in element B after transport is less than ROMULTI, no transport is done.
  - b. If the density of a specific material in element A after transport is less than ROMULTI, all of the mass of that material is transported to element B.
3. By default, the cut-off density is set for each material separately as 1.E-5 times the material reference density.

**ROSTR**

Default = Remark 3., SOL 700 only

Defines the minimum density for single-material Eulerian elements with shear strength.

**Format:**

PARAM,ROSTR,VALUE

**Example:**

PARAM,ROSTR,1.E-6

VALUE      Density cutoff. (Real &gt; 0.0)

**Remarks:**

1. Single-material Eulerian elements with shear strength with a density less than ROSTR are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than ROSTR, then no transport is done.
  - b. If the density of element A after transport is less than ROSTR, then all of the mass is transported to element B.
3. By default the cut-off density for Eulerian elements with shear strength is set to 1.E-5 times the material reference density.

**ROTSEKE**

Default = -1

Controls the printout of rotor energies in the .f06 file. For a model with only one rotor, the strain and kinetic energy percentages will always be 100%.

- 1 Rotor energies are printed in the .f06 file. (Default)
- 0 Rotor energies are not printed in the .f06 file.

## RSPECTRA

Default = -1

RSPECTRA = 0 requests that response spectra be calculated for transient analysis. See [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide* for a discussion of this capability. Response spectra will be calculated for any superelements or the residual structure for which other output requests are present in the same run. The requirements for the other output requests are also in [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide*. Any punch data produced is sent to the standard Nastran PUNCH file. Related parameters are TABID and RSPRINT.

## RSPRINT

Default = 0

RSPRINT controls the printout of tabulated values of response spectra. RSPRINT = -1 suppresses the printout. The related parameter is RSPECTRA.

## RSTTEMP

Default = NO

In SOL 106, PARAM,RSTTEMP,YES will cause the automatic restart to ignore changes to or additions of TEMPij Bulk Data entries. This is applicable to SOL 106 restart runs in which the temperature changes are only intended to affect the loading and not the material properties. The restart run also requires the use of the DBLOCATE FMS statement instead of the RESTART statement to reference the data base; e.g.,

```
assign run1=plate-run1.MASTER  
dbloc logical=run1
```

## S1, S1G, S1M

Default = -1

The MAXMIN Case Control command offers more features with much greater efficiency. PARAM,S1i,+1 requests the sorting and/or filtering of the element stresses selected on the DTI,INDTA entry. Stresses in the element coordinate systems (S1), at grid points (S1G), and/or in material coordinate systems (S1M) based on the parameters BIGER, NUMOUT, SRTOPT, and SRTELTYP may be requested. The S1G and S1M options also require the presence of PARAM,CURV,1.

| Parameter    | Quantity | Coordinate System | Location                              | Elements                                |
|--------------|----------|-------------------|---------------------------------------|---|
| $S1 \geq 0$  | Stresses | Element           | Element centers                       | CQUAD4,<br>CQUAD8,<br>CTRIA3,<br>CTRIA6 |
| $S1M \geq 0$ | Stresses | Material          | Element centers                       | CQUAD4,<br>CTRIA3                       |
| $S1G \geq 0$ | Stresses | Material          | Grid points to which elements connect | CQUAD4,<br>CTRIA3                       |

NUMOUT, in conjunction with BIGER, controls the amount of stress output.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1. Similarly, NUMOUT3 and BIGER3 are used for composite strength ratio.

1. NUMOUT = +N requests that N element stresses be printed (or punched) for each element type.
2. NUMOUT=0 outputs all elements in a group when one or more exceeds BIGER. Some of the elements will have stresses small than BIGER. This is conceptually the same as describing an element set in case control, and limiting output in this manner. Stress files obtained with element group filtering may be used for xy plotting and other postprocessor options with DMAP alters. By contrast, the stress file when NUMOUT = -2 is more discontinuous, and may not be used for xy plotting.
3. NUMOUT = 0 does not sort but filters according to BIGER by element group. In static analysis an element group is defined as all case control selected elements for a given load case for SORT1 output. For SORT2 output an element group is defined as the data for a given element type for all load cases. In transient analysis an element group is defined as all case control selected elements at a given time for SORT1 output. For SORT2 output an element group is defined as the data for a given element at all time steps. The element group option applies only to output types described above for PARAM,S1. This option is not available with output types selected by PARAMs S1G and S1M.
4. NUMOUT = -1 requests that stresses be sorted and only those stresses with an absolute value that is greater than BIGER will be output.
5. NUMOUT = -2 (the Default) does not sort but filters according to BIGER. Related parameters include BIGER, NOELOF, NOELOP, and NOGPF.
6. NUMOUT2 = -3 outputs the maximum value of Failure Index for each element.

BIGER controls the elements for which stresses will be printed. Elements with stresses that are smaller in absolute value than BIGER will not be output. The quantity tested is element type dependent. Related parameters include CURV, NUMOUT, S1, S1G, and S1M. SRTOPT controls the scanning option to be performed.

| SRTOPT Value | Description                             |
|--------------|---|
| 0            | Filter/sort on maximum magnitude.       |
| 1            | Filter/sort on minimum magnitude.       |
| 2            | Filter/sort on maximum algebraic value. |
| 3            | Filter/sort on minimum algebraic value. |

SRTELTYP controls the element type to be processed, as described in the following table.

| SRTELTYP Value | Description                                   |
|----------------|---|
| 0              | All element types will be processed.          |
| > 0            | Only element type SRTELTYP will be processed. |

NUMOUT1 and BIGER1 serve the same function as NUMOUT and BIGER except that they apply only to composite element stresses and do not require PARAM,S1i,+1.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1.

## S1AG,S1AM

Default = -1

See CURV.

## SBSPFORM

No Default

This parameter can be used to specify the Subspace Iteration method formulation for the structure-fluid coupled modes computation.

Subspace iteration method selects the formulation automatically based on the ratio of structure and fluid size and the existence of rigid body or constant pressure modes. But using this parameter, user can force one or other of the methods.

MF: Mobility-Flexibility formulation

SM: Stiffness-Mass formulation

## SCRSPEC

Default = -1 (SOLs 103 and 115 only)

SCRSPEC=0 requests that structural response be calculated for response spectra input in normal modes analysis. See [Response Spectrum Analysis](#) (Ch. 9) in the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of this capability. The scaled response calculations are made for elements and grid points of the

residual structure only. There exist two basic methods which are controlled by PARAM,CQC. The default (CQC=0) selects the traditional method. CQC > 0 selects a more recent method called the complete quadratic method of peak response combination—also called the CQC method. In both methods, the responses are summed with the ABS, SRSS, NRL, or NRLO convention, depending on the value of PARAM,OPTION. If the SRSS, NRL, or NRLO options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies meet the inequality  $f_{i+1} < \text{CLOSE} \cdot f_i$ . Both PARAM,OPTION and PARAM,CLOSE may be set in any subcase, allowing summation by several conventions in a single run.

In Version 70, the NRL option has been modified slightly to correspond to the NAVSEA-0908-LP-000-3010 specification. NRLO provides the V69 NRL.

PARAM,CQC,1 or 2 selects the complete quadratic method of peak response combination; also called the CQC method. PARAM,CQC,1 selects CQC method of response combination for sum across modes. PARAM,CQC,2 is same PARAM,CQC,1, but outputs the CQC coefficients for each mode ij pair and frequency and damping for each mode. The default is to consider all modes in the calculation but if only a subset of modes are of interest then a DTI,CQC,1,... Bulk Data entry may be used to specify a list of modes (by mode number) to retain for the CQC solution. When multiple excitation directions are specified, PARAM,CQC,1 or 2 specifies the summation across the modes and PARAM,OPTION specifies the summation across excitation directions. As with PARAM,OPTION and PARAM,CLOSE, when PARAM,CQC is used at the subcase level, each subcase can have a different value specified for OPTION, CLOSE and CQC. This allows comparisons to be made in the same run among summation across directions using ABS and SRSS methods while using the CQC method to sum across modes.

For the CQC method, PARAM,DIROUT,YES (Default = NO) outputs the responses combined across the modes for each separate excitation direction as well as the responses combined across modes and directions. This allows results to be assessed per excitation direction and for the total combined response.

PARAM,POST,-1 will write the additional directional responses to the op2 file for post processing with Patran. In this case, the directional responses will be labeled in the Patran Results menu with a TIME value corresponding to the excitation sequence number. For example, if the SUPORT entry specifies the following degrees of freedom to be excited: SUPORT,1,123 then direction 1 will be labeled with TIME value 1, direction 2 with TIME value 2, etc. PARAM,POST,-1 will also write the additional maximum response to the op2 file for post processing. In this case, the maximum response will be labeled in the Patran Results menu with a TIME value 4.

For the CQC method, PARAM,NEWMARK,YES (Default = NO) may be used in the case where 3 simultaneous excitation directions are defined and calculate the Newmark combinations across the excitations directions using the 40% rule. PARAM,PERCENT (Default = 40) may be used to specify a different percentage for Newmark combinations.

## SDCSV

Default = 0

Dictates what stability derivative information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM, SDUNIT, n. SDCSV has the following options which can be summed to select multiple options:

- 0 No output
- 1 Rigid aero
- 2 Rigid Splined
- 4 Elastic restrained
- 8 Elastic unrestrained
- 16 Intertial restrained
- 32 Intertial unrestrained

## SDRPOPT

Default = SDRP

SDRPOPT controls the storage of the principal stresses and strains in the stress and strain tables (OES1 and OSTR1 data blocks) in p-version analysis. By default, the principal stresses and strains are stored in the stress and strain tables to support postprocessing. PARAM,SDRPOPT,OFP requests that the principal stresses and strains are not stored in the tables. This can result in a significant reduction in disk space usage.

## SDUNIT,n

n SDUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of stability derivative results in SOL 144.

## SEKD

Default = RSONLY

Controls the calculation of the upstream superelement differential stiffness matrix in SOL 106.

- RSONLY Calculate the differential stiffness for the residual structure only.
- ALL Calculate and reduce the differential stiffness for all superelements and the residual structure.

## SEMAP, SEMAPOPT, SEMAPPRT

SEMAP Default = SEMAP

SEMAPOPT Default = 42

SEMAPPRT Default = 3

The superelement map (SEMAP table) contains several lists useful for determining how the program has partitioned superelement models. It is printed automatically each time this table is generated. It consists of three major parts:



|     |  |
|-----|--|
| GPM | The Grid Point Map contains a list of each grid point, its interior superelement placement, and the SEID of all grid points connected directly to it by elements. Three tables follow that summarize the connectivity between superelements, sorted on grid point sequence, SEID, and the number of connections. |
| ISM | The Individual Superelement Map lists the interior grid points, exterior grid and scalar points, elements, and time and storage space estimates for each superelement.   |
| SDT | The Superelement Definition Table contains the SEID of every superelement in the model, the processing order, and a pictorial representation of the superelement tree.   |

SEMAP, SEMAPOPT, and SEMAPPRT are used to control the amount of output that is printed and other special features. The possible values for SEMAP are shown in the following table.

| SEMAP Value     | Output and Application   |
|-----------------|--|
| SEMAP (Default) | ISM, SDT. The lengthy GPM is suppressed. This is the appropriate value for use after the model is stable and only minor changes are to be made.  |
| SEMAPALL        | GPM, ISM, SDT. All tables are printed. This value is useful on the initial debug run of a model and when making extensive modeling changes.  |
| SEMAPCON        | Only the summary tables of the GPM and the estimation data is output. This is a useful value when iterating to an economic partitioning scheme for large, complex models.  |
| SEMAPEST        | Only the estimation data is printed. This is useful when evaluating several alternative partitioning schemes.  |
| SEMAPPUN        | No output is printed. The exterior grid points of the superelement with a SEID that is input on SEMAPOPT are placed on a CSUPER entry image on the PUNCH file, allowing the superelement to be used as an external superelement. If SEMAPOPT > 0, the superelement entry is given an SSID of SEMAPOPT. If SEMAPOPT < 0, the exterior points listed are those of the residual structure, but the CSUPER entry is given an SSID of  SEMAPOPT . |

Other special features are available with parameters SEMAPOPT and SEMAPPRT. They are fully described under parameters OPT1 and OPT2 in the description of the TABPRT module in the [MSC Nastran DMAP Programmer's Guide](#).

If the default value of SEMAP is used, the other two parameters may be used to further refine this output, as described in [MSC Nastran DMAP Programmer's Guide](#) under the TABPRT module description. The printing of the SEMAP table can be avoided by the use of PARAM,SEMAPPRT,-1.

## SENSUOO

Default = NO

By default, in dynamic sensitivity analysis in SOL 200, displacements at the o-set due to pseudo-loads do not include any effect due to inertia and damping. If PARAM,SENSUOO,YES is specified then these effects will be computed in a quasi-static manner and included in the sensitivity analysis.

## SEP1XOVR

Default = 0

The old and new location of moved shell grid points are printed if SEP1XOVR = 16. When the RSSCON shell-to-solid element connector is used. By default, the moved shell grid points are not printed, SEP1XOVR = 0. See the description of PARAM,TOLRSC for more details.

## SEQOUT

Default = 0

See OLDSEQ.

## SERST

Default = AUTO

By default, all restarts are considered automatic (see [Restart Procedures in the MSC Nastran Reference Guide](#)). If none of the following Case Control commands are entered, then SEALL=ALL is the default action: SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL.

These commands may be used to partition the analysis into several runs. By default, the restart will proceed in automatic fashion for each command, regenerating only that data that is affected by modifications in the Bulk Data and Case Control or changes in upstream superelements. If the user wishes to overwrite the data, even if it is not affected by modifications to the data, then PARAM,SERST,MANUAL must be entered.

With PARAM,SERST,AUTO or MANUAL, all superelements will be processed through Phase 0 (see [Superelement Analysis in the MSC Nastran Reference Guide](#)). This phase includes execution of the sequencer module (SEQP), initial superelement processing (SEP1), and initial geometry processing (GP1 and GP2) modules, which can result in significant CPU overhead. If this overhead is not desired, then PARAM,SERST,SEMI will limit Phase 0 and Phase 1 to only those superelements specified on the SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL Case Control commands. If none of these commands is entered, then execution will skip Phase 0 and 1.

In the modal solution sequences (SOLs 110, 111, 112, 145, 146, and 200), the modes of the residual structure are automatically computed in Phase 2 if any SE-type command (e.g., SEMG=n) is requested for the residual structure. If PARAM,SERST,SEMI and no SE-type command is specified for the residual structure, then, by default, its modes will not be recomputed. This logic is intended for restarts from SOL 103 into one of the modal solutions. If, however, the modes have not already been computed or need to be recomputed, then PARAM,SERST,RSMDS must be specified to force the calculation of the residual structure modes.

If PARAM,SERST,SEDR is specified, then Phases 0, 1, and 2 will be skipped. This option is intended for data recovery (Phase 3) runs only.

The options of SEMI, RSMDS, and SEDR are intended for models that are defined on more than one database; i.e., superelements are defined on separate databases (multiple MASTER DBsets) and processed in separate runs. Also, with this technique, databases are attached with the DBLOCATE File Management statement rather than the RESTART File Management statement. In general, these options are not recommended because they disable the automatic restart capability, which compromises the database integrity.

## SESDAMP

Default = NO

### PARAM,SESDAMP,YES

Modal damping is calculated for superelements if PARAM,SESDAMP,YES is specified. An SDAMPING Case Control command that selects a TABDMP1 Bulk Data entry must also be specified in the desired superelement's subcase. By default, modal damping is added to viscous damping (B). If you insert PARAM,KDAMP,-1 (or PARAM,KDAMPFL,-1 for fluid superelements) then modal damping will be added to structural damping (K4). The SESDAMP parameter needs to be specified as NO for an assembly run.

### PARAM,SESDAMP,AUG

By its nature, classical modal damping in superelements is computed only for the component modes. If fixed boundary or mixed boundary (some fixed, some free) methods are used to compute the component modes, no damping is computed for the constraint modes, which can lead to answer differences when comparing the same model with and without superelements. An enhanced modal damping method has been developed to account for damping on the constraint modes. To use the enhanced damping method instead of the classical method, define "PARAM,SESDAMP,AUG". When this enhanced damping method is defined, the user may choose how the damping factor for the constraint modes is determined via the parameter TDAMP.

## SESEF

Default = -1 (SOLs 103 and 115 only)

If SESEF = 0 in superelement normal modes analysis, the fraction of total strain energy for a superelement in each of the system's modes is output in the vector SESEFA for tip superelements and in SESEFG for nontip superelements. If SESEF = 1, strain energy fractions are output, and expansion of the eigenvectors from a-set size to g-set is branched over for tip superelements. If SESEF = -1 (the default value), no strain energy fractions are computed.

Output requests must be present in order for strain energy fractions to be calculated. If SESEF = 1, no other output results for tip superelements.

## SHIFT1

Default = -1.234

The negative shift used when computing massless mechanism constraint modes with PARAM,MMMETH,OLD. For very stiff model (1000. hz for the first flexible mode), consider using a larger value. See also [MECHFIX, 898](#) and [MMMETH, 908](#).

## SHLDAMP

Default = SAME

If SAME, then structural damping is obtained from MID1 material of PSHELL. If DIFF or any value not equal to SAME, each MIDi field of the PSHELL will have its own structural damping. See Remark 5. of the PSHELL.

## SIGMA

Default = 0.0

The radiant heat flux is proportional to

$$\text{SIGMA} \cdot (T_{grid} + T_{ABS})^4$$

where SIGMA is the Stefan-Boltzmann constant,  $T_{grid}$  is the temperature at a grid point, and  $T_{ABS}$  is the scale factor for absolute temperature and may be specified by PARAM,TABS. These parameters must be given in units consistent with the rest of the data in the model. The value for SIGMA in SI units is

$$5.67 \times 10^{-8} \text{ watts/m}^2 \text{K}^4$$

The default value causes radiant heat effects to be discarded.

## SKINOUT

Default = NONE

Request that sets of grid and element lists be output for both the fluid and structure at the fluid-structure interface.

- |       |  |
|-------|--|
| NONE  | Requests no output of sets.                |
| PUNCH | Requests set output to .pch only.          |
| PRINT | Requests set output to .f06 only.          |
| ALL   | Requests set output to both .pch and .f06. |

See the Case Control command FLSPOUT as an alternative selection.

## SKPAMP

Default = 0

For SOLs 145, 146, and 200, SKPAMP = -1 suppresses all unsteady aerodynamic calculations. The automatic restart performs a similar function without this parameter. Specifying it ensures suppression of the calculations, regardless of the determination of the automatic restart.

## SLOOPID

Default = 0 (SOL 129 and 159 only)

In a nonlinear transient analysis (SOLs 129 and 159) restart, SLOOPID identifies the initial conditioning previous nonlinear analysis run (SOLs 106 and 153 respectively). Setting SLOOPID greater than 0 will cause SOLs 129 and 159 to start from the static deformed position.

## SMALLQ

Default = 0.0

By default Nastran removes unused superelement q-set degrees-of-freedom from the residual structure solution set. Set this parameter to a small value (e.g., 1.0E-10) if you do not want unused superelement q-set degrees-of-freedom removed.

## SNORM

Default = 20.0

SNORM > 0.0 requests the generation of unique grid point normals for adjacent shell elements (see [Figure 6-2](#)). Unique grid point normals are generated for the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements. The grid point normal is the average of the local normals from all adjacent shell elements including CQUAD8 and CTRIA6 elements. If grid point normals are present, they are used in all element calculations of the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

| SNORM | Tolerance in Degrees  |
|-------|---|
| > 0.0 | Unique grid point normals are generated if each angle between the grid point normal and each local normal of the adjacent shell elements is smaller than SNORM. SNORM Bulk Data entries overwrite a generated normal. |
| = 0.0 | The generation of grid point normals is turned off. The user can define normals with the SNORM Bulk Data entry.   |
| < 0.0 | Grid point normals are not generated. SNORM Bulk Data entries are ignored.  |

**Caution:** If the grid shown in [Figure 6-2](#) is located on a symmetric half model boundary and, hence, Shell 2 is not present, you may attain the same result as a full model by specifying the normal direction with the SNORM Bulk Data entry.

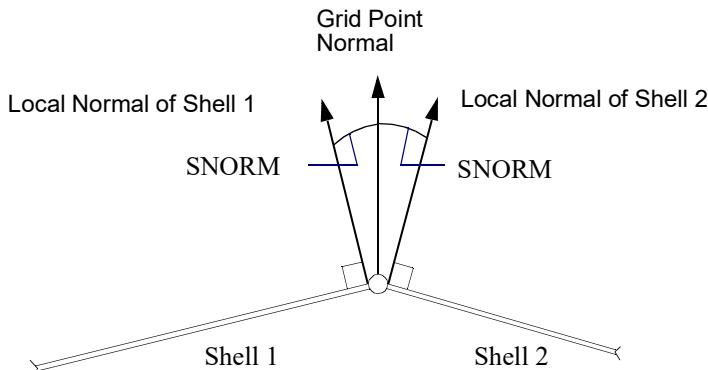


Figure 6-2 Unique Grid Point Normal for Adjacent Shell Elements

## **SNORMPRT**

Default = -1

PARAM,SNORMPRT,>0 writes the grid point normals of the model in the basic coordinate system to the .f06 and/or .pch files.

| SNORMPRT | Switch to Print Out Normals                         |
|----------|---|
| $\leq 0$ | No output   |
| 1        | Print out to the punch file (.pch)                  |
| 2        | Print out to the print file (.f06)                  |
| 3        | Print out to the punch (.pch) and print file (.f06) |

## **SOFTEXIT**

Default = NO

In SOL 200, if soft convergence is achieved during optimization, before completing the maximum number of design iterations, the user may request an exit with PARAM,SOFTEXIT,YES.

## **SOLADJC**

Default = 0

PARAM SOLADJC indicates if adjoint solution vectors are to be calculated during the analysis:

- 1      Do not calculate adjoint solution vectors during the analysis. Any required adjoint solution vectors will be computed during sensitivity analysis.
- $\geq 0$       For ANALYSIS=DFREQ, the adjoint vectors will be computed during the solution if:
  1. All frequency response DRESP1 entries are grid responses.
  2. Each subcase has the same set of excitation frequencies.
  3. The number of degrees-of-freedom referenced on DRESP1 entries < (number of independent design variables + number of type-2 properties + number of spawned nonlinear beam library properties) x (number of subcases.)
  4. PARAM AUTOADJ=YES (Default)
- 999999      For ANALYSIS=MFREQ, the adjoint vectors will be computed during the solution if the four items above are satisfied and SOLADJC=999999. SOLADJC=999999 cannot be used in conjunction with FREQ3, FREQ4 or FREQ5 entries to specify excitation frequencies.

## SOLID

Default = 0

SOLID also allows several models to be stored in the same graphics database created by PARAM,POST,0.

## SPARSEDM

Default = YES

See SPARSEDR.

## SPARSEDR

Default = YES

SPARSEDR=YES limits the data recovery matrix calculations to recover only those grid points specified on SET commands referenced by grid point output requests (DISP, SPCF, etc.) or those points connected to elements specified on element output requests (STRESS, FORCE, etc.). In SOL 200, the design model and design responses are also taken into consideration for determining which grid points are needed for data recovery.

SPARSEDM=YES is for SOL 200 and takes further advantage of small design models/responses during the adjoint or pseudo-load sensitivity calculations.

These methods take advantage of very small output requests, and/or small design models in SOL 200, for large models resulting in significant CPU and disk space savings.

If, however, the output requests and/or the size of the design model in SOL 200 require the calculation of the solution over a large enough percentage of degrees-of-freedom, then it is more efficient to compute the solution at all grid points. The user PARAMeter SPDRRAT (Default75) and SPDMRAT (Default60) specifies this percentage.

The sparse data recovery method is not supported in Aeroelastic (SOL 146), Flutter (SOLs 145 and 200), Complex Eigenvalue (SOLs 107, 110, 145, and 200), Nonlinear (SOLs 106, 129, 153, 159, and 400) and Cyclic Symmetry Analysis (SOLs 114, 115, and 118).

The sparse data recovery method is deactivated when the following Case Control commands are specified: EKE, ESE, EDE, and CMSENRGY.

PARAM,DDRMM is ignored under PARAM,SPARSED,YES. To restore the data recovery solution process to pre-V2004 methods insert PARAM,SPARSED,NO.

## **SPARSEPH**

Default = YES

The default selects the very efficient sparse data recovery method during eigenvector data recovery in DOMAIN SOLVER ACMS (PARTOPT=DOF). This method uses the same process for eigenvector data recovery as is requested by PARAM,SPARSED,YES in other dynamic solution sequences like SOLs 107 through 112.

SPARSEPH will be automatically changed to NO if ACPOWER or INTENSITY calculations are requested. These calculations require all eigenvector DOF to be computed.

PARAM,SPARSEPH,NO forces full data recovery on all degrees-of-freedom and can be significantly less efficient if data recovery is required at only a few degrees-of-freedom. See related user parameters [MDOTM](#), [897](#) and [SPARSED, 996](#).

## **SPDRRAT**

Default = 75

See SPARSED.

## **SPCGEN**

Default = NO

SPCGEN is analogous to and described by the PUNCH keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,SPCGEN.

## **SPDMRAT**

Default = 60

See SPARSED.

## **SQSETID**

Default = 99000001

See [AUTOQSET, 794](#).

## **SRCOMPS**

Default = NO

SRCOMPS controls the computation and printout of ply strength ratios. If SRCOMPS=YES, ply strength ratios are output for composite elements that have failure indices requested.

## **SRTELTYP**

Default = 0

See S1, S1G, S1M.

## **SROPT**

Default = 0

See S1, S1G, S1M.

## **START**

Default = 0

See OLDSEQ.

## **STEPFCT**

Real, Default = 0.666, SOL 700 only. The scale factor has an effect on the time step computation of Lagrangian and Euler elements.

## **STEPFCTL**

Real > 0, Default = 0.9, SOL 700 only.

Defines a scale factor to be used on the internally calculated time step. When both of STEPFC and STEPFCTL are defined, STEPFCTL will be used for Lagrangian elements and STEPFC will be used for Euler elements.

## **STIME**

Default = 0.0 (SOLs 109, 112, 129 and 159 only)

In restarts from previous transient analysis runs, the user provides STIME =  $t_N$  where  $t_N$  is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from  $t_N$  as though the original run contained the new subcase data.

In SOLs 109 and 112 restarts from previous SOLs 109 and 112 runs, STIME is used to specify the proper starting time of the restart run. If STIME exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to STIME. In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any time earlier than that. The program informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run.

In SOLs 109 and 112 restarts, the user must ensure that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run. The user may, however, specify different TSTEP and DLOAD requests in Case Control and also different TSTEP and dynamic loading entries in the Bulk Data compared to the previous run. The loading and the results output from the restart run will start from the new starting time.

**Notes:**

1. The responsibility for ensuring that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run is left to the user; the program does not check for this condition. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.
2. Restarts in SOLs 109 and 112 using STIME > 0.0 are supported **only** for applied loads, **not** for SPC/SPCD enforced motion. If this usage is violated, the program terminates the execution with an appropriate fatal message.

**STRUCTMP**

Replaced by a keyword on the FLSPOUT Case Control command.

**SUBCASID**

Default = 0

PARAM,SUBCASID,n where n is greater than zero, specifies that the restart proceeds from SUBCASE n in nonlinear static analysis, SOL 106. SUBCASID is an alternative to SUBID and is recommended over SUBID which indicates the subcase sequence number.

**SUBID**

Default = 1

In SOL 106 by default, the restart proceeds from the last LOOPID in the last subcase. SUBID may be used to specify an earlier subcase by specifying the sequential number (for SEID = 0) of the subcase. In SOLs 106 and 153, PARAM,LOOPID may also be specified for an earlier LOOPID. SUBCASID is an alternative to SUBID and is recommended over SUBID. See [Restarts for Nonlinear Static Analysis, 684](#) in Appendix C: Nonlinear Analysis in *MSC Nastran Reference Guide* for further discussion.

**SUPAERO**

Default = ZONA

If SUPAERO=ZONA, then the ZONA51 code is used for supersonic aerodynamic calculations. If SUPAERO=CPM, then the CPM method is used. If ZONA51 is not available at a particular installation, PARAM,SUPAERO,CPM must be specified to avoid a fatal error when performing supersonic aerodynamic analyses. Only one supersonic aerodynamics method can be selected in a given run.

**Note:**

CPM is only supported for planar configurations.

**SUPDOF**

Default = 0

Digits 1 through 6 indicate which of the six rigid body degrees of freedom are supported for a Solution 144 run. For example, SUPDOF=35 indicates that the plunge (3) and pitch (5) degrees of freedom are supported. This parameter is optional. If omitted, Nastran will compute these data using a numerical method.

**SUPER**

Default =  $\begin{cases} 0 & \text{(nonsuperelement sequences)} \\ -1 & \text{(superelement sequences)} \end{cases}$

See OLDSEQ.

**TABID**

Default = 2

TABID controls the punch output for response spectra. See [Appendix C page 597](#) in *MSC Nastran Reference Guide*. A related parameter is RSPECTRA.

**TABS**

Default = 0.0

TABS is used to convert units of the temperature input ( $^{\circ}\text{F}$  or  $^{\circ}\text{C}$ ) to the absolute temperature ( $^{\circ}\text{R}$  or  $^{\circ}\text{K}$ ). Specify:

PARAM,TABS,273.16      When Celsius is used.

PARAM,TABS,459.69      When Fahrenheit is used.

Refer to the Bulk Data entry, [CREEP, 1628](#) for a creep analysis with SOLs 106 or 153. Refer to PARAM,SIGMA for heat transfer analysis.

**TDAMP**

PARAM,TDAMP,Real

When using enhanced modal damping for superelements (see PARAM,SESDAMP,AUG ), the user may choose how the damping factor for the constraint modes is determined.

If TDAMP=0.0 (the default), the damping factor is averaged over all the damping coefficients defined on TABDMP1.

If TDAMP is less than 0.0, the damping factor from TABDMP1 associated with the lowest natural frequency is used.

If TDAMP is positive, the damping factor defined by TDAMP is used directly.

**TESTNEG**

Default =  $\begin{cases} -2 & \text{for Newtons method} \\ -1 & \text{for Arc-length method and SOLs 101-112,115,118,187, and} \\ & 200 \end{cases}$

In nonlinear static analysis (SOLs 106 and 153), and differential stiffness generation with preload (SOLs 101-112, 115, 118, 187, and 200 with STATSUB) this parameter specifies the action to take when negative terms are encountered on the factor diagonal of matrix decomposition. Negative terms indicate that the differential

stiffness has introduced a structural instability. The instability may be real (structural buckling) or mathematical (the current iteration appears unstable, but a stable solution exists).

| TESTNEG | Results   |
|---------|---|
| -1      | Stop if negative terms occur.   |
| 1 or 0  | Continue if negative terms occur.   |
| -2      | If negative terms exist, do not use differential stiffness.   |
| 2       | Do not use differential stiffness.  |
| 3       | Use differential stiffness with preload even if negative terms exist (SOLs 101-112, 115, 118, 187, and 200 with STATSUB only) |

## TFSYMFAC

Default = 1.D-08

TFSYMFAC specifies the tolerance at which matrices generated via the TFL Case Control command are treated as anti-symmetric in the solution process. The default will not symmetrize the matrix even if the tolerance is not exceeded. Whereas, if TFSYMFAC is negative then the absolute values is used as the tolerance and the matrix will be symmetrized if the tolerance is exceeded. If TFSYMFAC is 0.D0 then the symmetry is not checked.

## TINY

Default = 1.E-3

Any elements with strain energy that is less than a TINY percentage of the total strain energy for any superelement will not be printed or made available for postprocessing by MSC Patran or other programs. TINY may also be used to suppress the printing of small numbers in the constraint check matrix [ $E_{mh}$ ] described in [Geometry Processing in SubDMAP PHASE0 in the MSC Nastran Reference Guide](#).

## TOLRSC

Default = 0.05

When the RSSCON shell-to-solid element connector is used, the connecting grid points of the shell element are moved on to the solid face if the grid points are close enough. The tolerable distance of the shell grid point to the solid edge or face is  $\epsilon \cdot h$  where  $h$  is the height of the solid edge; see the sample figure below. The relative tolerance is user modifiable using the parameter.

PARAM,TOLRSC, $\epsilon$

The default for the relative tolerance is  $\epsilon = 0.05$ . Rigid body invariance is satisfied with double-precision accuracy if the shell grid points are adjusted.

## TOPOCONV

Default = 80

Parameter TOPOCONV is applicable only to ESLNRO topology optimization tasks. It sets a lower bound for the percentage of the design variables whose maximum relative changes are within the tolerance specified by CONVDV on the DOPTPRM entry.

By default, when more than 80% of the design variables show their maximum relative changes are within CONVDV, the job will be terminated.

## TRMBIM

Default = PHYSICAL

TRMBIM is used to define the formulation used for reduced impedance matrix, RIM, for all trim components requested under TRIMGRP case control commands. It is default to 'PHYSICAL' which will generate RIM in physical coordinates. The other valid option for TRMBIM are:

1. 'MODAL' which will produce RIM in modal coordinates.
2. 'DATACHK' which perform dry run without producing Reduce Impedance Matrix or results. Job will be terminated afterwards.

It is important to note that data recovery for trim components are supported only under 'PARAM,TRMBIM,PHYSICAL' and 'PARAM,PEMNPART,1 (default)' for all trim components. In addition, SOL 200 supports for trim components is limited to following conditions,

1. Design model on structure and/or fluid model only (not the trim/PEM material).
2. Design responses/constraints on responses of structure and/or fluid only (not the trim/PEM material).

## TSTATIC

Default = -1 (SOLs 129, 159, 400 and 600 only)

If TSTATIC = 1, a static solution may be obtained while ignoring inertial and damping forces. For SOLs 129 and 159, this option is available only with the adaptive time-stepping method (see METHOD = "ADAPT" on the Bulk Data entry, [TSTEPNL, 3286](#)). For SOL 600, TSTATIC=1 may only be used for the default auto step iteration procedure.

## UGASC

SOL 700 only,

Real. Defines a value for the universal gas constant.

1. Must be used if the molar weight is used on a GAS option of the AIRBAG entry, or if molar gas fractions are given on an INFLFRAC entry.
2. Specify only one universal gas constant per problem.
3. In IS units, R equals  $8.3145 \text{ J mol}^{-1} \text{ K}^{-1}$ .

Using the tonne, mm, system of units R has a value of  $8314.5 \text{ tonne mm}^2 \text{ s}^{-2} \text{ mol}^{-1} \text{ K}^{-1}$ .

In imperial units, R equals  $1.9859 \text{ Btu lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$  or  $1545.3 \text{ ft lbf lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$

## UNSYMF

Default = NO

In SOL 106, nonlinear statics, PARAM,UNSYMF,YES is required to include damping effects in the calculation of complex eigenvalues. See PARAM,NMLOOP.

## UNSYMKB

Character, default = 'NO'

In modal solutions or real eigenvalue solutions, if the stiffness or damping matrix is not symmetric, the run will normally terminate with a FATAL error message. Setting UNSYMKB to 'YES' will allow the run to continue.

## UPDTBSH

Default = NO

UPDTBSH controls the update of boundary shapes generated by auxiliary boundary model analysis in SOL 200. By default, the auxiliary boundary models and shapes are generated only once at the initial design cycle and will not be updated in subsequent cycles even if the shape of the primary model is changing. PARAM,UPDTBSH,YES requests that the auxiliary models and shapes are updated and reanalyzed at every cycle.

## USETPRT

Default = -1

USETPRT controls the tabular printout of the structural degree-of-freedom sets. See [Degree-of-Freedom Sets, 1111](#). To print aerodynamic degree-of-freedom sets, please see [OPGEOM](#).

| Sequence | Print               | USETPRT |
|----------|---------------------|---------|
| None     | None (Default)      | -1      |
|          | Row sort only       | 0       |
| Internal | Column sort only    | 1       |
|          | Row and Column sort | 2       |
| External | Row sort only       | 10      |
|          | Column sort only    | 11      |
|          | Row and Column sort | 12      |

The degrees-of-freedom can be listed in ascending order according to their internal or external sequence number, but not both. The external sequence number is the grid, scalar, or extra point identification number. The internal sequence number is the number assigned after resequencing (see PARAM,OLDSEQ).

The row sort is not recommended in p-version analysis because large integers are generated for hierarchical grid point identification numbers and they will be truncated.

For a given sequence there are two types of tables that may be printed: row sort and column sort. For row sort, a table is printed for each set selected by USETSEL. Here is an example of row sort (USETPRT = 0 or 10):

| U S E T D E F I N I T I O N T A B L E ( I N T E R N A L S E Q U E N C E , R O W S O R T ) |     |     |     |     |     |     |     |     |      |
|---|-----|-----|-----|-----|-----|-----|-----|-----|------|
|   | A   |     |     |     |     |     |     |     |      |
| -1-   | -2- | -3- | -4- | -5- | -6- | -7- | -8- | -9- | -10- |
| 1=  | 2-1 | 2-2 |     |     |     |     |     |     |      |

For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT=1 or 11):

| U S E T D E F I N I T I O N T A B L E ( I N T E R N A L S E Q U E N C E , C O L U M N S O R T ) |     |     |     |     |     |     |    |    |   |   |   |   |   |   |   |   |   |   |
|---|-----|-----|-----|-----|-----|-----|----|----|---|---|---|---|---|---|---|---|---|---|
| EXT   | GP. | DOF | INT | DOF | INT | GP. | SB | SG | L | A | F | N | G | R | O | S | M | E |
| ---   |     |     |     |     |     |     |    |    |   |   |   |   |   |   |   |   |   |   |
|   | 1   | - 1 |     | 1-  |     | 1 G |    | 1  |   |   |   | 1 | 1 |   |   |   |   | 1 |
|   |     | - 2 |     | 2-  |     |     |    | 2  |   |   |   | 2 | 2 |   |   |   |   | 2 |
|   |     | - 3 |     | 3-  |     |     |    |    | 1 |   |   | 3 | 3 |   |   |   |   | 3 |
|   |     | - 4 |     | 4-  |     |     |    |    | 2 |   |   | 4 | 4 |   |   |   |   | 4 |
|   |     | - 5 |     | 5-  |     |     |    |    | 3 |   |   | 5 | 5 |   |   |   |   | 5 |
|   |     | - 6 |     | 6-  |     |     |    |    | 4 |   |   | 6 | 6 |   |   |   |   | 6 |

## USETSEL

Default = 0

USETSEL specifies the sets which will be printed in the row sort (USETPRT = 0 or 10). In order to select specific sets to be printed, you must sum their corresponding decimal equivalent numbers. For example, sets A, L, and R are selected with USETSEL=128+256+8=392.

| USETSEL | Sets Printed   |
|---------|--|
| -1      | All sets as defined in <a href="#">Degree-of-Freedom Sets, 1111</a> .        |
| 0       | Mutually exclusive sets only; i.e., sets M, SB, SG, O, Q, R, C, B, E, and A. |

## USETSTR1

Input-character-Default' ‘

USETSTR1 through USETSTR4 specifies the sets that will be printed by the specification of parameters USETPRT and USETSEL. Any set in [Degree-of-Freedom Sets, 1111](#) may be specified. A “:” is used as a separator. In the following example, the m-set (degrees-of-freedom eliminated by multipoint constraints) and s-set (degrees-of-freedom eliminated by single point constraints) are specified.

### Example:

PARAM,USETSTR1,M:S

## VARPHI

Default =  $\pi/4$  (0.78539816)

Feature angle for detection of singular geometric features (sharp edges or corners) in the mesh during adaptive mesh refinement.

When adaptive meshing is requested (see Case Control command, [HADAPT \(Case\), 417](#) and Bulk Data entry, [HADPLT, 2126](#)), singular geometric features such as sharp edges or sharp corners must be detected. To this end face outward normals  $N_1, N_2$  of each pair of adjacent mesh faces and the edge oriented tangents  $T_1, T_2$  of each pair of adjacent mesh edges are computed (see [Figure 6-3](#)). If the angle between  $N_1$  and  $N_2$  for mesh faces, or between  $T_1$  and  $T_2$  for mesh edges is bigger than the feature angle  $\varphi$ , then the common edge or vertex will be considered a splitting edge or vertex where surfaces or lines are broken and a singular geometric feature is defined.

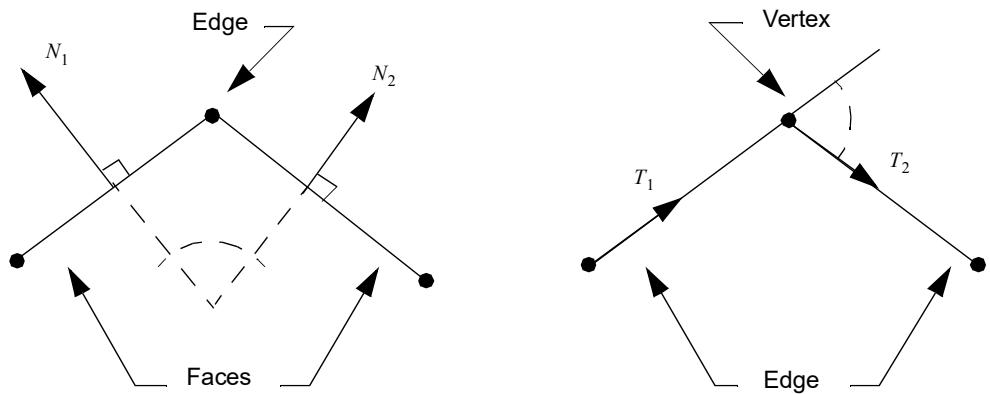


Figure 6-3

Mesh faces and elements are preprocessed to ensure consistent orientation and that the appropriate sign of face normals and edge tangents will be accounted for during the computation of their mutual angle.

## VELCUT

Default = 1.E-6, SOL 700 only

Defines the minimum velocity in Eulerian meshes.

### Format:

PARAM,VELCUT,VALUE

### Example:

PARAM,VELCUT,1.0E-6

VALUE            Minimum velocity. (Real > 0.0)

**Remark:**

Any velocity less than VELCUT is set to zero. It is mainly used to eliminate harmless but annoying small values of velocity caused by round-off error and numerical dispersion.

**VMOPT**

Default = 0

By default (VMOPT=0), virtual mass is included after the component modes are computed. Virtual mass is not taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=1, then the virtual mass will be included in the mass matrix at the same time as all other mass elements. In other words, the component modes will reflect the virtual mass. Virtual mass is taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=2, the modes of the structure or component without the fluid are computed first ("dry" modes). The fluid effects are added to the modal basis during the residual flexibility computation to produce the "wet" modes for the component. Both eigenvalue tables are printed, allowing comparison of the dry and wet modes. The wet modes are used in modal dynamic analysis. Cost savings result from the dense Virtual Mass (VM) matrix being kept out when computing dry modes in the physical basis. Its presence can increase memory and computation times by an order of magnitude. The VM is added only to the smaller generalized basis used in Residual Flexibility Computations. The approximations introduced by this approach are generally small due to the homogeneous nature of the fluid. It is the preferred method when the number of wetted elements exceeds several hundred, for reasons of efficiency. It is to be noted that MAX normalization for eigenvectors with PARAM,VMOPT,2 may produce incorrect results. Therefore, MAX normalization is not recommended when using PARAM,VMOPT,2.

If VMOPT is not equal to 0, 1 or 2, then no virtual mass is computed.

**VREF**

Default = 1.0

In modal flutter analysis, the velocities are divided by VREF to convert units or to compute flutter indices.

**VUELJUMP, VUGJUMP**

Default = 1000

Specifies the separation in identification numbers for display elements and grid points generated in p-version analysis. The defaults are sufficient for a 9 9 display element mesh.

Identification numbers for display elements and grid points start with 10001001 and 201001001, respectively. For example, by default the identification numbers for the display elements of the first p-element will be numbered 100001001 through 100002000 and the second p-element 100002001 through 100003000, etc.

**VUBEAM, VUHEXA, VUPENTA, VUQUAD4, VUTETRA, VUTRIA3**

Default = VUBEAM

Default = VUHEXA

Default = VUPENTA

Default = VUQUAD4

Default = VUTETRA

These parameters are used in p-version analysis to specify the names of the display elements in the data recovery output tables; such as those created by the VUGRID Case Control command and PARAM,POST. They should be used if your postprocessor does not recognize display elements. For example, PARAM,VUHEXA,CHEXA renames the display element VUHEXA to “CHEXA” in the output files.

## WHIRLOPT

Default = FWD

Control forward whirl or backward whirl analysis for SYNC option in SOL 107 and SOL 108 for analysis in rotating reference frame using the ROTOR entry.

Usage:

PARAM, WHIRLOPT, FWD: For forward whirl analysis

PARAM, WHIRLOPT, BWD: For backward analysis

## WMODAL

Default = NO

In Modal Transient analysis (SOL 112), PARAM, WMODAL, YES will automatically produce accurate conversion at all frequencies using the modal frequencies based on the eigenvalues as shown below. This relieves the user of trying to determine the “best” value for (PARAM, W3 and W4). Note, use of WMODAL does not suppress user specified values of W3 and W4.

$$\begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix} [\Phi_{dq}^\tau] (G[K_{dd}^1] + [K_{dd}^4]) [\Phi_{dq}] \begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix}$$

Note the CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery do not currently take into account the WMODAL conversion in SOL112.

## WRBEAMB

Integer, Default = 0

- 0 Write equivalent radius for all beams (see PARAM, BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc’s GEOMETRY values for beam type elements.
- 1 Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.

## WTMAS600

Integer, Default = 1

Determines whether GRAV entries will be scaled by WTMAS600 or not for SOL 600 only.

- 0      The value of A (4th field) on the GRAV entry will not be scaled by WTMAS600.
- 1      The value of A (4th field) on the GRAV entry will be scaled by WTMAS600.

### Remarks:

1. This parameter is available starting with the MD Nastran 2010.
2. Prior to MD Nastran 2010, WTMAS600=1 for all analyses.

## WTMASS

Default = 1.0

The terms of the structural mass matrix are multiplied by the value of WTMAS600 when they are generated. In coupled fluid-structure analysis WTMAS600 is applied to the structural portion of the model only. WTMAS600 applies to MFLUID entries but it is not recommended for use in hydroelastic problems.

## W3, W4, W3FL, W4FL

Default = 0.0

The damping matrix for transient analysis is assembled from the equation:

$$[B_{dd}] = [B_{dd}^1] + [B_{dd}^2] + \frac{G}{W3}[K_{dd}^1] + \frac{1}{W4}[K_{dd}^4]$$

In coupled fluid-structure analysis, W3 and W4 are applied to structural portion of the model and W3FL and W4FL to the fluid portion of the model. The default values of 0.0 for W3, W4, W3FL, and W4FL cause the  $[K_{dd}^1]$  and  $[K_{dd}^4]$  items to be ignored in the damping matrix, regardless of the presence of the PARAM, G or GFL or  $[K_{dd}^4]$ .  $[K_{dd}^1]$  is the stiffness.

$[K_{dd}^4]$  is the structural damping and is created when GE is specified on the MATi entries.

$[K_{dd}^1]$  is the stiffness. The units of W3, W4, W3FL, and W4FL are radians per unit time. (See [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS in the MSC Nastran Reference Guide](#) for further discussion.)

In SOLs 129 and 159, W4 may vary between subcases. However, the linear portion of the model uses only the W4 value from the first subcase and the values in the subsequent subcases are applied to the nonlinear portion of the model.

Parameters G, W3, W4 are case control selectable by either the PARAM entries in Case Control, or RSDAMP or SEDAMP appearing in Case Control pointing to a bulk data DAMPING entry. If both DAMPING type entries and PARAM entries occur in a subcase, the DAMPING entries take precedence.

When there is no Parameters G, W3, W4 or DAMPING reference in the first Case Control, the Bulk Data specification (if any) will be used. Subsequent Case Control specification will override the Bulk Data specification and Case Control rules apply to subsequent specifications.

The CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL109 and SOL112.

Linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL400 with ANALYSIS=NLTTRAN. These elements behave in SOL400 with ANALYSIS=NLTTRAN in a linear fashion when PARAM, LGDISP, -1 (Default) and there are no PBUSHT referring to the CBUSH or PELAST entries referring to the CELAS1, CELAS2, CELAS3.

For linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements in SOL400 with ANALYSIS=NLTTRAN, SOL400 considers Case Control specification of G, W3, or W4 as a data recovery boundary condition change and each STEP*i* for the linear elements will pick up the appropriate G, W3, and W4 values.

Any of these elements in SOL400 with ANALYSIS=NLTTRAN that are nonlinear only use the standard force output data recovery F=Ku(t) since nonlinear elements compute only the unbalanced stiffness force.

In SOL129 or SOL159 linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements (determined as in SOL400 above) should never have G, W3, or W4 selected in the Case Control as the linear force recovery will be incorrect as only the G, W3, and W4 of the first subcase is used.

In SOL129, the FUSE option of the CBUSH should never be used if GE or B is specified for the fusing CBUSH. This is because, unlike SOL400, SOL129 does not update the B or K4 matrix after the CBUSH fuses. Thus remnant CBUSH forces due to damping will be present.

## WR3, WR4, WRH

Default = 0.0, no rotor damping or circulation terms.

Specifies “average” excitation frequency for calculation of rotor damping and circulation terms. See “Equations Used in Analyses” on page 192 of the *MSC Nastran 2004 Release Guide* for equations.

## XFACT

Default=1.0

Additional factor to determine upper bound for Q-set reorthogonalization subspace, for ACMS VERSION=NEW (module ACMS1).

## XFACTX

Default=2.0

Multiplier applied to user's maximum frequency to determine maximum frequency for reorthogonalization for ACMS VERSION=NEW (module ACMS1).

## XFLAG

Default = 0

By default (XFLAG = 0), when temperature loads and element deformations are present, the element strain energy for the linear elements is calculated using the following equation:

$$E = \frac{1}{2} u^T K_e u - u^T P_{et}$$

where  $u$  is the deformation,  $K_e$  is the element stiffness and  $P_{et}$  is the element load vector for temperature differences and element deformations. If XFLAG is set to 2, the element strain energy for linear elements is calculated using the following equation:

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et}$$

The latter formula is the same strain energy calculation used for nonlinear elements.

When requesting Case Control commands ESE or EDE and executing SOL 108 or SOL 111, if Nastran detects frequency dependent elements (CBUSH if referred to by PBUSH/PBUSHT; CELAS1, CELAS3 if referred to by PELAS/PELAST; CDAMP1, CDAMP3 if referred to by PDAMP/PDAMPT), Nastran will automatically remove the entire class of CBUSH, or CELAS1, or CELAS3, or CDAMP1, or CDAMP3 elements, whether or not all of a class are frequency dependent, from the residual (SE0) energy calculations because frequency dependent elements are not fully implemented in the ESE/EDE calculations and may result in wrong answers. USER INFORMATION MESSAGE 5245 will be issued for each class removed. If the user wishes to override this removal, set XFLAG to 32. If the user wants XFLAG=2 and XFLAG=33, then set XFLAG=34.

## XYMPCH

Default = NO

The XYPILOT/XYPUNCH commands do not support SOL103 type mode shapes. XYMPCH when set to a value PARAM, XYMPCH, YES allows the XYPUNCH command XYPUNCH, DISP, RESPONSE / ... to punch without selective selection SOL103 mode shapes as SOL101 type displacements. For example:

```
$  
METHOD = 10  
SET 100 = 6, 7, 13  
PARAM, XYMPCH, YES  
DISP=100  
MODES=20 $  
...  
OUTPUT (XYPILOT)  
XYPUNCH DISP RESPONSE / 6 (T3) $  
XYPUNCH DISP RESPONSE / 7 (T3) $  
XYPUNCH DISP RESPONSE / 13 (T2) $
```

A more modern solution is to include in a standard SOL103 run the Nastran entry

NASTRAN HDF5=0

With for example DISP(PLOT)=ALL

And extract the resulting mode shapes from the resulting HDF5 Data Base.

Then the PARAM, XYMPCH, and MODES=n is not needed nor is any of the OUTPU(XYPILOT) commands.

## XYUNIT, n

- n XYUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of design optimization results and design sensitivity data in comma separated value format for use in a spreadsheet in SOL 200 or the<sup>c</sup> storage of trim results in SOL 144.

## ZROCMAS

Default = NO

When performing component modal synthesis with free or mixed boundary conditions, the c-set mass is normally included during the calculation of the component modes. If the component has large masses on the c-set degrees-of-freedom, or if the user requests too many modes for the component, the c-set residual flexibility will become singular. This causes a failure of the component reduction. The singularity may be avoided by setting ZROCMAS to YES, which will exclude the c-set mass when calculating the component modes.

## ZROVEC

Default=1.E-06

ZROVEC specifies the tolerance at which a residual vector is not linearly independent and subsequently removed from the residual vector computation process.

## Parameter Applicability Tables

[Table 6-1](#), [Table 6-2](#), [Table 6-3](#), [Table 6-4](#), and [Table 6-5](#) list parameter applicability to the solution sequences (SOLs 101 through 114).

[Table 6-6](#), [Table 6-7](#), [Table 6-8](#), [Table 6-9](#), [Table 6-10](#), [Table 6-11](#), [Table 6-12](#), [Table 6-13](#), [Table 6-14](#), [Table 6-15](#), and [Table 6-16](#) list parameter applicability to the solution sequences (SOLs 115 through 700).

- B      Must be specified in the Bulk Data Section only.
- E      May be specified in either the Bulk Data and/or Case Control Section.
- C      Must be specified in the Case Control Section only.

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| ACEXTMD        |   |     |     |     |     |     |     |     | B   |     |     |
| ACEXTSET       |   |     |     |     |     |     |     |     | B   |     |     |
| ACOUT          | B   |     |     |     | E   | E   | E   | E   | E   | E   |     |
| ACOWEAK        |   |     |     |     |     |     |     |     | B   |     |     |
| ACSYM          |   |     |     |     | B   |     |     |     | B   |     |     |
| ADJMETH        |   |     |     |     |     |     |     |     |     |     |     |
| ADMPOST        |   |     |     |     |     |     |     |     | E   | E   |     |
| ADPCON         |   |     |     | E   |     |     |     |     |     |     |     |
| ADSTAT         |   |     |     |     |     | B   |     |     |     | B   |     |
| AERODOF        |   |     |     |     |     |     |     |     |     |     |     |
| AESDISC        |   |     |     |     |     |     |     |     |     |     |     |
| AESMAXIT       |   |     |     |     |     |     |     |     |     |     |     |
| AESMETH        |   |     |     |     |     |     |     |     |     |     |     |
| AESRNDM        |   |     |     |     |     |     |     |     |     |     |     |
| AESTOL         |   |     |     |     |     |     |     |     |     |     |     |
| ALPHA1         |   |     |     |     | B   | B   | B   | B   | B   | B   |     |
| ALPHA2         |   |     |     |     | B   | B   | B   | B   | B   | B   |     |
| ALPHA1FL       |   |     |     |     |     |     |     |     |     |     |     |
| ALPHA2FL       |   |     |     |     |     |     |     |     |     |     |     |
| ALTRED         | B   |     | B   |     |     |     |     |     |     |     | B   |
| ALTSHAPE       | B   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| ARBMASP        |   |     |     |     |     |     |     |     |     |     |     |
| ARBMFEM        |   |     |     |     |     |     |     |     |     |     |     |
| ARBMPMS        |   |     |     |     |     |     |     |     |     |     |     |
| ARBMS          |   |     |     |     |     |     |     |     |     |     |     |
| ARBMS          |   |     |     |     |     |     |     |     |     |     |     |
| ARBMSTYP       |   |     |     |     |     |     |     |     |     |     |     |
| ARF            |   |     |     |     |     |     |     |     |     |     |     |
| ARS            |   |     |     |     |     |     |     |     |     |     |     |
| ASCOUP         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| ASING          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| AUNITS         |   |     |     |     |     |     |     |     |     |     |     |
| AUTOADJ        |   |     |     |     |     |     |     |     |     |     |     |

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| AUTOGOUT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| AUTOMSET       | E   | E   | E   |     | E   | E   | E   | E   | E   | E   | E   |
| AUTOQSET       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| AUTOSPC        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| AUTOSPCR       |   |     |     | E   |     |     |     |     |     |     |     |
| AUTOSPRT       |   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |
| BAILOUT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| BEAMBEA        |   |     |     |     |     |     |     |     |     |     |     |
| BEIGRED        |   |     |     | E   | E   | E   | E   | E   | E   | E   |     |
| BETA           |   |     |     |     |     |     |     |     |     |     |     |
| BIGER          | E   | E   | E   |     |     |     |     |     |     |     | E   |
| BIGER1         | E   | E   | E   |     |     |     |     |     |     |     | E   |
| BIGER2         | E   | E   | E   |     |     |     |     |     |     |     | E   |
| BUCKLE         |   |     |     | E   |     |     |     |     |     |     |     |
|                |   |     |     |     | E   |     |     |     |     |     |     |
| CASPIV         | E   |     |     |     |     |     |     |     |     |     |     |
| CB1, CB2       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CDIF           |   |     |     |     |     |     |     |     |     |     |     |
| CFDIAGP        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CFRANDEL       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CHECKOUT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CK1, CK2, CK3  | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CLOSE          |   | B   |     |     |     |     |     |     |     |     |     |
| CM1, CM2       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CONFAC         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| COUPMASS       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CP1, CP2       | E   |     | E   | E   |     | E   | E   |     | E   | E   | E   |
| CQC            |   | B   |     |     |     |     |     |     |     |     |     |
| CURV           | E   | E   | E   |     |     |     |     |     |     |     | E   |
| CURVPLOT       | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| CWDIAGP        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CWRANDEL       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBALL          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| DBCCONV        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBC DIAG       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBCOVWRT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBDICT         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| DBDN           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| DBDRPRJ        | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| DBDRVER        | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| DBEXT          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBRCV          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DBUP           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| DDRMM          |   |     |     |     |     |     |     |     | B   | B   |     |
| DESPCH         |   |     |     |     |     |     |     |     |     |     |     |
| DESPCH1        |   |     |     |     |     |     |     |     |     |     |     |
| DFREQ          |   |     |     |     |     | B   |     |     | B   |     |     |
| DOPT           | E   | E   | E   |     |     |     |     |     |     |     | E   |
| DPEPS          |   |     |     |     |     |     |     |     |     |     |     |
| DSNOKD         |   |     | B   |     |     |     |     |     |     |     |     |
| DSZERO         | B   | B   | B   |     |     |     |     |     |     |     |     |
| DYNSPCF        |   | E   |     |     | E   | E   | E   | E   | E   | E   | E   |
| EIGFILT        |   | B   |     |     |     |     |     |     | B   |     |     |
| ENFMETH        |   |     |     |     | E   | E   |     |     | E   | E   |     |
| ENFMOTN        |   |     |     |     | E   | E   |     |     | E   | E   |     |
| ERPC           |   |     |     |     |     |     |     |     |     |     |     |
| ERPREFDB       |   |     |     |     |     |     |     |     |     |     |     |
| ERPRHO         |   |     |     |     |     |     |     |     |     |     |     |
| ERRLRF         |   |     |     |     |     |     |     |     |     |     |     |
| ERROR          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| ESLFSAV        |   |     |     |     |     |     |     |     |     |     |     |

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| ESLMOVE        |   |     |     |     |     |     |     |     |     |     |     |
| ESLRCF         |   |     |     |     |     |     |     |     |     |     |     |
| EST            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| EXTDR          | B   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| EXTDROUT       | B   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| EXTDRUNT       | B   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| EXTOUT         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| EXTRCV         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| EXTUNIT        | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| FACTOR         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| FBATOLR        |   |     |     |     |     | E   |     |     | E   |     |     |
| FBLEND         |   |     |     |     |     |     |     |     |     |     |     |
| FIXEDB         | E   | E   |     |     |     |     |     |     |     |     |     |
| FKSYMFAC       |   |     |     |     | E   |     |     |     |     |     |     |
| FLUIDSE        |   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| FOLLOWK        |   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| FRQDEPO        |   |     |     |     |     | B   |     |     |     | B   |     |
| FULLSEDR       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| FZERO          |   | E   |     | E   | E   | E   | E   | E   | E   | E   | E   |
| G              |   |     |     |     | B   | B   | B   | B   | B   | B   |     |
| GEOMU          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| GFL            |   |     |     |     | B   | B   | B   | B   | B   | B   |     |
| GPECT          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| GRDPNT         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| GUSTAERO       |   |     |     |     |     |     |     |     |     |     |     |
| GYROAVG        |   |     |     |     |     | E   |     |     | E   |     |     |
| HEATSTAT       | B   |     |     |     |     |     |     |     |     |     |     |
| HFREQ          |   |     |     |     |     |     |     | B   | B   | B   |     |
| HFREQFL        |   |     |     |     |     |     |     | B   | B   | B   |     |
| HTOCITS        |   |     |     |     | E   |     |     |     |     |     |     |
| HTOCPRT        |   |     |     |     | E   |     |     |     |     |     |     |
| HTOCTOL        |   |     |     |     | E   |     |     |     |     |     |     |
| IFP            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| INREL          | E   |     | E   |     |     |     |     |     |     |     |     |
| IRES           | E   |     | E   | E   |     | E   | E   |     | E   | E   | E   |
| ITAPE          | B   | B   | B   |     |     |     |     |     |     |     |     |
| IUNIT          | B   | B   | B   |     |     |     |     |     |     |     |     |
| KDAMP          |   |     |     |     |     |     |     | B   | B   | B   |     |
| KDAMPFL        |   |     |     |     |     |     |     | B   | B   | B   |     |
| KDIAG          |   |     |     | E   |     |     |     |     |     |     |     |
| K6ROT          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| LANGLE         |   |     |     | B   |     |     |     |     |     |     |     |
| LFREQ          |   |     |     |     |     |     |     | B   | B   | B   |     |
| LFREQFL        |   |     |     |     |     |     |     | B   | B   | B   |     |

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| LGDISP         |   |     |     | E   |     |     |     |     |     |     |     |
| LMFACT         | B   | B   | B   |     |     |     |     |     |     |     |     |
| LMODES         |   | E   |     |     |     |     |     | B   | B   | B   |     |
| LMODESFL       |   |     |     |     |     |     |     | B   | B   | B   |     |
| LOADU          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| LOOPID         |   |     |     | E   |     |     |     |     |     |     |     |
| MACH           |   |     |     |     |     |     |     |     |     |     |     |
| MAXLP          |   |     |     |     |     |     |     |     |     |     |     |
| MAXRATIO       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| MDK4OPT        |   | B   |     |     |     |     |     | B   | B   | B   |     |
| MDOPT14        |   | B   |     |     |     |     |     | B   | B   | B   |     |
| MDOTM          |   | B   |     |     |     |     |     | B   | B   | B   |     |
| MDOTMFAC       |   | B   |     |     |     |     |     | B   | B   | B   |     |
| MDREDOPT       |   | B   |     |     |     |     |     | B   | B   | B   |     |
| MESH           | B   |     |     |     |     |     |     |     |     |     |     |
| METHCMRS       |   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| MHRED          |   | E   |     |     | E   | E   | E   | E   | E   | E   |     |

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| MINIGOA        | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| MODACC         |   |     |     |     |     |     |     | B   | B   | B   |     |
| MODEL          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| MPCX           | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| MPTUNIT        |   |     |     |     |     |     |     |     |     |     |     |
| MXICODE0       |   |     |     |     |     |     |     |     |     |     |     |
| NASPRT         |   |     |     |     |     |     |     |     |     |     |     |
| NDAMP          |   |     |     |     |     |     |     |     |     |     |     |
| NEWSET         | B   |     |     |     |     |     |     |     |     |     |     |
| NHPLUS         |   |     |     |     |     |     |     |     |     |     |     |
| NINTPTS        | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NLAYERS        |   |     |     |     | E   |     |     |     |     |     |     |
| NLHTLS         |   |     |     |     | E   |     |     |     |     |     |     |
| NLHTOL         |   |     |     |     |     |     |     |     |     |     |     |
| NLHTWK         |   |     |     |     |     |     |     |     |     |     |     |
| NLMAX          |   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |
| NLMIN          |   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |
| NMLOOP         |   |     |     | E   |     |     |     |     |     |     |     |
| NOCOMPS        | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| NOELOF         | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NOELOP         | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NOGPF          | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NOMSGSTR       | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NONCUP         |   |     |     |     |     |     |     |     |     |     | B   |
| NQSET          |   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| NLTOL          |   |     |     | B   |     |     |     |     |     |     |     |
| NUMOUT         | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NUMOUT1        | E   | E   | E   |     |     |     |     |     |     |     | E   |
| NUMOUT2        | E   | E   | E   |     |     |     |     |     |     |     | E   |
| OCMP           | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| OEE            | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| OEF            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OEFX           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| OELMOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OELMSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OES            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OESE           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OESX           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OG             | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| OGEM           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OGPF           | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| OGPS           | E   | E   | E   |     |     |     | E   |     |     | E   | E   |
| OGRDOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OGRDSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OLDSEQ         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| OMAXR          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OMID           | E   | E   | E   |     |     | E   | E   | E   | E   | E   | E   |
| OMGLVL         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OPCHSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OPG            | E   |     | E   | E   |     | E   | E   |     | E   | E   | E   |
| OPGEOM         |   |     |     |     |     |     |     |     |     |     |     |

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| OPGPKG         |   |     |     |     |     |     |     |     |     |     |     |
| OPPHIB         |   |     |     |     |     |     |     |     |     |     |     |
| OPPHIPA        |   |     |     |     |     |     |     |     |     |     |     |
| OPTEXIT        |   |     |     |     |     |     |     |     |     |     |     |
| OPTION         |   | B   |     |     |     |     |     |     |     |     |     |
| OQG            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OSETELE        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OSETGRD        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OSWELM         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| OSWPPT         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| OUG            | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OUGCORD        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OUGSPLIT       |   |     |     | E   |     | E   | E   |     | E   | E   |     |
| OUMU           |   | E   |     |     |     |     |     |     |     |     |     |
| OUNIT1         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OUNIT2         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OUTOPT         | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| PACINFIID      |   |     |     |     |     | B   |     |     | B   |     |     |
| PATPLUS        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PATVER         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PCOMPRM        | B   |     |     |     |     | B   | B   |     | B   | B   |     |
| PDRMSG         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PEDGEP         | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| PENFN          | B   | B   | B   |     |     |     |     |     |     |     |     |
| PLTMSG         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| POST           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| POSTEXT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| POSTU          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PREFDB         |   |     |     |     |     |     | E   | E   | E   | E   |     |
| PRINT          |   |     |     |     |     |     |     |     |     |     |     |
| PROUT          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRPA           | E   |     |     | E   |     |     |     |     |     |     |     |
| PRPHIVZ        |   | E   |     |     | E   | E   | E   | E   | E   | E   | E   |
| PRPJ           | E   |     | E   |     |     | E   | E   |     | E   | E   | E   |
| PRTBGPDT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTCSTM        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTEQXIN       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTGPDT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTGPL         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTGPTT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTMAXIM       | E   | E   | E   |     |     |     |     |     |     |     |     |
| PRTMGG         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PRTPG          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| PRTRESLT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| PVALINIT       | B   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| Q              |   |     |     |     |     |     |     |     |     |     |     |
| RBSETPRT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| RESLOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| RHOCP          |   |     |     |     |     |     |     |     |     |     |     |
| ROTSEKE        |   |     |     |     | E   |     |     | E   |     |     |     |
| RSPECTRA       |   |     |     |     |     |     | E   |     |     |     | E   |
| RSPRINT        |   |     |     |     |     |     | E   |     |     |     | E   |
| RSTTEMP        |   |     |     | E   |     |     |     |     |     |     |     |
| S1             | E   | E   | E   | E   |     |     | E   |     |     | E   | E   |
| S1A            | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| S1AG           | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| S1AM           | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| S1G            | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| S1M            | E   | E   | E   | E   |     |     |     |     |     |     | E   |
| SCRSPEC        |   | B   |     |     |     |     |     |     |     |     |     |
| SEMAP          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SEMAPOPT       | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SEMAPPRT       | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SENSUOO        |   |     |     |     |     |     |     |     |     |     |     |
| SEP1XOVR       | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SEQOUT         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SERST          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SESDAMP        |   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| SESEF          |   | E   |     |     |     |     |     |     |     |     |     |

Table 6-5      PARAMeter Names in SOLs 101 Through 114 Part 5

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 114 |
| SHLDAMP        |   |     |     |     |     |     |     |     | E   |     |
| SIGMA          |   |     |     |     |     |     |     |     |     |     |
| SKINOUT        |   | E   |     |     | E   | E   | E   | E   | E   | E   |
| SKPAMP         |   |     |     |     |     |     |     |     |     |     |
| SLOOPID        |   |     |     |     |     |     |     |     |     |     |
| SMALLQ         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SNORM          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SNORMPRT       | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SOFTEXIT       |   |     |     |     |     |     |     |     |     |     |
| SOLADJC        |   |     |     |     |     |     |     |     |     |     |
| SOLID          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| SPARSEDM       | E   | E   | E   |     |     | E   | E   |     | E   | E   |
| SPARSEDRL      | E   | E   | E   |     |     | E   | E   |     | E   | E   |
| SPARSEPH       |   | E   |     |     |     |     |     | E   | E   | E   |
| SPDDMMAT       |   |     |     |     |     |     |     |     |     |     |
| SPDRRAT        | E   | E   | E   |     |     | E   | E   |     | E   | E   |
| SQSETID        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| SRTELTYPE      | E   | E   | E   |     |     |     |     |     |     | E   |
| SROPT          | E   | E   | E   |     |     |     |     |     |     | E   |
| START          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| STIME          |   |     |     |     |     |     | E   |     |     | E   |
| SUBCASID       |   |     |     | E   |     |     |     |     |     |     |
| SUBID          |   |     |     | E   |     |     |     |     |     |     |
| SUPAERO        |   |     |     |     |     |     |     |     |     |     |
| SUPER          | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| TABID          |   |     |     |     |     |     | E   |     |     | E   |
| TABS           |   |     |     |     |     |     |     |     |     |     |
| TESTNEG        |   |     |     | E   |     |     |     |     |     |     |
| TFSYMFAC       |   |     |     |     | E   | E   | E   | E   | E   | E   |
| TINY           | E   | E   | E   |     |     |     |     |     |     | E   |
| TOLRSC         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |
| TSTATIC        |   |     |     |     |     |     |     |     |     |     |

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5 (continued)

| PARAMeter Name | Solution Sequence Numbers (101 through 114) |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 101   | 103 | 105 | 106 | 107 | 108 | 109 | 110 | 111 | 112 | 114 |
| UNSYMF         |   |     |     | B   |     |     |     |     |     |     |     |
| UNSYMKB        |   | B   |     |     | B   | B   | B   | B   | B   | B   |     |
| UPDTBSH        |   |     |     |     |     |     |     |     |     |     |     |
| USETPRT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| USETSEL        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| VARPHI         | C   |     |     |     |     |     |     |     |     |     |     |
| VMOPT          |   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| VREF           |   |     |     |     |     | B   |     |     |     |     | B   |
| VUBEAM         | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUELJUMP       | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUGJUMP        | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUHEXA         | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUPENTA        | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUQUAD4        | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| VUTETRA        | E   | E   |     |     | E   | E   | E   | E   | E   | E   |     |
| WTMASS         | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| W3             |   |     |     |     |     |     | E   |     |     |     | E   |
| W3FL           |   |     |     |     |     |     | E   |     |     |     | E   |
| W4             |   |     |     |     |     |     | E   |     |     |     | E   |
| W4FL           |   |     |     |     |     |     | E   |     |     |     | E   |
| XFACT          |   | B   |     |     |     |     |     |     |     | B   |     |
| XFACTX         |   | B   |     |     |     |     |     |     |     | B   |     |
| XFLAG          | E   |     | E   |     |     |     |     |     |     |     | E   |
| XYUNIT         |   |     |     |     |     |     |     |     |     |     |     |
| ZROCMAS        |   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |
| ZROVEC         |   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| ACEXTMTD       |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| ACEXTSET       |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| ACOUT          |   |     |     |     |     |     |     |     |     | E   |     |     |     |
| ACOWEAK        |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| ACSYM          |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| ADJMETH        |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| ADPCON         |   |     |     | E   |     |     |     |     | E   | E   |     | E   |     |
| ADSTAT         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| AERODOF        |   |     |     | E   |     | E   |     | E   |     |     | E   |     |     |
| AESDISC        |   |     |     | B   |     |     |     |     |     |     | B   |     |     |
| AESMAXIT       |   |     |     | B   |     |     |     |     |     |     | B   |     |     |
| AESMETH        |   |     |     | B   |     |     |     |     |     |     | B   |     |     |
| AESRNDM        |   |     |     | B   |     |     |     |     |     |     | B   |     |     |
| AESTOL         |   |     |     | B   |     |     |     |     |     |     | B   |     |     |
| ALPHA1         |   | B   | E   |     | B   | B   | B   | B   | B   | B   | E   |     |     |
| ALPHA2         |   | B   | E   |     | B   | B   | B   | B   | B   | B   | E   |     |     |
| ALPHA1FL       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ALPHA2FL       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ALTRED         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARBMASP        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARBMFEM        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARBMPMS        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARBMS          |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARBMSTYP       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARF            |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ARS            |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ASCOUP         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ASING          |   |     |     |     |     |     |     |     |     |     |     |     |     |
| AUNITS         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| AUTOADJ        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| AUTOGOUT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| AUTOMSET       | E   | E   | E   |     | E   | E   | E   |     | E   |     | E   |     | E   |

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| AUTOQSET       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |     |
| AUTOSPC        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| AUTOSPCR       |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| AUTOSPRT       | E   |     | E   | E   | E   | E   | E   |     | E   |     | E   |     |     |
| BAILOUT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| BEAMBEA        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| BEIGRED        |   |     | E   | E   |     | E   | E   |     |     | E   | E   |     |     |
| BETA           |   |     |     | E   |     |     |     |     | E   |     |     |     |     |
| BIGER          | E   | E   |     |     | E   |     |     | E   |     | E   |     |     |     |
| BIGER1         | E   | E   |     |     | E   |     |     | E   |     | E   |     |     |     |
| BIGER2         | E   | E   |     |     | E   |     |     | E   |     | E   |     |     |     |
| BUCKLE         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| CASIEMA        |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| CASPIV         |   |     |     |     |     |     |     |     |     | E   | E   |     |     |
| CB1, CB2       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CDIF           |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| CFDIAGP        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CFRANDEL       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CHECKOUT       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CK1, CK2, CK3  | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| CLOSE          | B   |     |     |     |     |     |     |     |     |     |     |     |     |
| CM1, CM2       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CONFAC         | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |     |
| COUPMASS       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CP1, CP2       |   | E   | E   | E   | E   |     | E   | E   | E   | E   | E   |     |     |
| CQC            | E   |     |     |     |     |     |     |     |     |     |     |     |     |
| CURV           | E   | E   |     |     | E   |     |     | E   |     | E   |     |     |     |
| CURVPLOT       | E   | E   |     |     | E   |     |     |     |     |     | E   | E   |     |
| CWDIAGP        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| CWRANDEL       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBALL          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBCCONV        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |

Table 6-7      PARAMeter Names in SOLs 115 Through 700 Part 2

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| DBCDIAG        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBCOVWRT       | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBDICT         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |     |
| DBDRPRJ        | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |     |
| DBDRVER        | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |     |
| DBDN           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBEXT          | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBRCV          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DBUP           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| DDRMM          |   |     |     |     |     |     | B   |     |     | E   |     |     |     |
| DEACTEL        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| DECLUMP        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| DESPCH         |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| DESPCH1        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| DFREQ          |   |     | B   |     |     |     | B   |     |     |     | B   |     |     |
| DOPPT          | E   |     |     |     | E   |     |     | E   |     |     | E   |     |     |
| DPEPS          |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| DPHFLG         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| DSNOKD         |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| DSZERO         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| DV3PASS        |   |     |     |     |     |     |     |     |     |     | B   |     |     |

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| DYNSPCF        | E   |     | E   |     |     | E   | E   |     |     | E   |     |     |     |
| ENFMETH        |   |     |     |     |     |     | E   |     |     | E   |     |     |     |
| ENFMOTN        |   |     |     |     |     |     | E   |     |     | E   |     |     |     |
| EPPRT          |   |     |     |     |     |     |     |     |     |     |     |     |     |
| EPSILONT       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| EPZERO         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ERPC           |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ERPREFDB       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ERPRHO         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ERPRLF         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ERROR          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| ESLFSAV        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLLCOMP       |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLMOVE        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLMPC1        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLOPTEX       |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLPRT         |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLPRT1        |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| ESLRCF         |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| EULBND         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| EULSTRES       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| EXTOUT         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| EXTRCV         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| EXTUNIT        | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| FACTOR         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| FBATOLR        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| FBLEND         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| FIXEDB         |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| FKSYMFAC       |   |     |     |     |     |     |     |     |     |     | E   |     |     |
| FLEXINCR       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| FLUIDMP        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| FLUIDSE        |   |     |     |     |     |     |     |     |     |     | B   |     |     |

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| FMULTI         |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| FOLLOWK        | E   | E   |     |     |     |     |     |     |     | E   | E   | B   |     |
| FRQDEPO        |   |     |     |     |     |     | B   |     |     | B   |     |     |     |
| FULLSEDR       | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| FZERO          |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| G              |   |     | B   | E   |     | B   | B   |     | B   | B   | B   |     |     |
| GFL            |   |     |     |     |     |     | B   |     |     |     | B   |     |     |
| GEOMU          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| GFL            |   |     |     |     |     |     | B   |     |     |     | B   |     |     |
| GPECT          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| GRADMESH       |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| GRDPNT         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |

Table 6-9 PARAMeter Names in SOLs 115 Through 700 Part 4

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| GUSTAERO       |   |     |     |     | B   | B   |     |     | B   |     |     |     |     |
| GYROAVG        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| HEATCMD        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| HEATSTAT       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| HFREQ          |   |     |     |     | B   | B   |     |     | B   |     |     |     |     |
| HFREQFL        |   |     |     |     |     | B   |     |     | B   |     |     |     |     |
| HTOCITS        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| HTOCPRT        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| HTOCTOL        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| IFP            | E   |     | E   | E   | E   | E   | E   | E   | E   | E   |     |     |     |
| INREL          |   |     |     |     |     |     |     |     |     | E   |     |     |     |
| INRLM          |   |     |     |     |     |     |     |     |     |     |     |     |     |
| IRES           |   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |     |
| ITAPE          |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| IUNIT          |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| KDAMP          |   |     |     |     | B   |     |     |     |     | B   |     |     |     |
| KDAMPFL        |   |     |     |     |     | B   |     |     |     | B   |     |     |     |
| KDIAG          |   |     |     |     |     |     |     | E   |     |     | E   |     |     |
| K6ROT          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| LANGLE         |   |     | B   |     |     |     |     | B   | B   |     |     | B   |     |
| LDSUM          |   |     |     | X   |     |     |     |     |     | X   |     |     |     |
| LFREQ          |   |     |     |     | B   | B   |     |     |     | B   |     |     |     |
| LFREQFL        |   |     |     |     |     | B   |     |     |     | B   |     |     |     |
| LGDISP         |   |     | E   |     |     |     |     | E   | E   |     | E   | B   |     |
| LMFACT         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| LMODES         |   |     |     |     | B   | B   |     |     |     | B   |     |     |     |
| LMODESFL       |   |     |     |     |     | B   |     |     |     | B   |     |     |     |
| LOADU          | E   |     | E   | E   |     |     |     | E   | E   | E   | E   |     |     |
| LOOPID         |   |     |     | E   |     |     |     | E   | E   |     | E   |     |     |
| MACH           |   |     |     |     |     | B   |     |     |     |     |     |     |     |
| MALIAS02       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MALIAS03       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-9      PARAMeter Names in SOLs 115 Through 700 Part 4 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MARALPHA       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARAUTO        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARBK105       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARBK106       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARC3901       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARC3D         |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MARC4401       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCASUM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCAUTO       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCASEL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCBEAM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCBUG        |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MARCBUSK       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCCBAR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCCENT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCCON2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCCON3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCCPY        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDEF        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDILT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDIS2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDIS3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDIS4       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCDMIG       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCEKND       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCEXIT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCFEAT,N     |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCFIL        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCFRIC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MARCGAPP       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCGLUE       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCHOST       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCIAMN       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCITER       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCL001       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCLOWE       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCLUMP       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCMATT       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCMEM        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCMNF        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCMPII       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCND99       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCNOER       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCOFFT       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCONLY       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCONTF       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCOOCC       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCOPT        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCOTIM       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPARR       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPENT       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPINN       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPPOS       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPPOST      |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPR99       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPRES       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPRN        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPRNG       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPRNR       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPROG       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MARCPTH        |   |     |     |     |     |     |     |     |     |     | B   |     |     |

Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MARCRBAL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCRBAR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCRBE2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCRBE3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCREVR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCRIGD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCRUN        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSAME       |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MARCSCLR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSETS       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSETT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSINC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSIZ3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSIZ4       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSIZ5       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSIZ6       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSLHT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSOLV       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCPGCC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSTIFF      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCSUMY       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCT16        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCT19        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTABL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTEDF       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTEDN       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTEMP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTETT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTIEC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTOL        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTOTD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 |
| MARCTOTL       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTOTT       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTUBE       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCTVL        |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCUSUB       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCVERS       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCWDIS       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCWELD       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARCWIND       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARELSTO       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARFACEA       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARFACEB       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARFATAL       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARGPFOR       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARIBOOC       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARIProj       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARMPCHK       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARMPCID       |   |     |     |     |     |     |     |     |     |     | C   |     |
| MARNOSET       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARNOT16       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARPLANE       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARPROCS       |   |     |     |     |     |     |     |     |     |     |     |     |
| MARRBAR2       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARROUTT       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARSHRII       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MARUPDAT       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MAXIREVV       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MAXLP          |   |     | E   |     |     |     |     |     | E   |     |     |     |
| MAXRATIO       | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| MBENDCAP       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MCNLPARM       |   |     |     |     |     |     |     |     |     |     | B   |     |
| MCON2D3D       |   |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MCORDUPD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MCSSHLC        |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MCSSHORR       |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MDAREAMD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MDK4OPT        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MDOPT14        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MDOTM          |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MDOTMFAC       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MDREDOPT       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| MDUMLOAD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MESH           |   |     |     |     |     |     |     | B   | B   | B   |     |     |     |
| METHCMRS       |   | E   | E   |     |     | E   | E   |     | E   | E   |     |     |     |
| MEXTRNOD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MEXTSEE        |   |     |     |     |     |     |     |     |     |     |     | C   |     |
| MFASTCMP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MFORCOR1       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MFSKIPPP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MGAPINIT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MGLUETOL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MHEATSHL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MHEATUNT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MHOUBOLT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MHRED          | E   |     | E   | E   | E   | E   | E   |     |     | E   |     |     |     |
| MICRO          |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| MINCLDSB       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINIGOA        | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| MINRECCC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINSOUTT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINVASHF       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINVCITR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINVCSHF       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINVCTOL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MINFMAX        |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MINVNMOD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MLSTRAIN       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MLSTRAN2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MMEMDETT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MNASTLDS       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MODACC         |   |     |     |     |     | B   | B   |     |     |     |     |     |     |
| MODEL          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| MOFFCORE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MOP2TITL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MOUTRCOR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MPCX           | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| MPERMPRT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MPTUNIT        |   |     |     |     |     |     | E   |     |     |     |     |     |     |
| MRAFFLOR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRAFFLOT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRAFFLOW       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRALIAS        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRBE2SNG       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRBEAMB        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRBEPARAM      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRBIGMEM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRBUKMTTH      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRC2DADD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCKBODY       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCKLOAD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCONADD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCONTAB       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCONSE        |   |     |     |     |     |     |     |     |     |     |     | C   |     |

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MRCNRES        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCCONVER      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCORDS        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCWANGL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRCWELD        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRDELTTT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRDISCMB       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRDYNOLD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MREIGMTH       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MREL1103       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRELRB         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRELSPCD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRENUELE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRENUGRD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRENUMBR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRENUMMT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRESTALL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRESULTS       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MREVPLST       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFINITE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFOLLO2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFOLLOW       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFOLLOW1      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFOLLOW3      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRFOLLOW4      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRGAPUSE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRHYPMID       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRITTYPE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMAT8A3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMAT8E3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMAT8N1       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMAT8N3       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMAXISZ       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MRMAXNUM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMEMSUM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRMTXKGG       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRNOCOR        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRORINTS       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPARALL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRELAST        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPBUSHT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MRPIDCHK       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPLLOAD4      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPLLOAD4R     |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPLSUPD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPREFER       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPSHCMP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRPSHELL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRRELNOD       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRRRCFILE      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRSETNAM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRSPAWN2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRSPLINE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRSPRING       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRT16STP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTABLIM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTABLS1       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTABLS2       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTABPRC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTFINAL       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTIMING       |   |     |     |     |     |     |     |     |     |     |     | B   |     |

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| MRTTHREAD      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRTSHEAR       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MRUSDATA       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSOLMEM        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPCCCHK       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPCKEEP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPEEDCB       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPEEDCW       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPEEDOU       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPEEDP4       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSPEEDSE       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSOLMEM, MBYTE |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MSTFBEAM       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MTABINGNR      |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MTABLD1M       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MTABLD1T       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MTEMPDWN       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MTLD1SID       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MULRFORC       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MUMPSOLV       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MUSBKEEP       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MWINQUOT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MWLDGSGA       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| MXLAGM1        |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| NASPRT         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| NDAMP          |   |     | E   |     |     |     |     |     |     | E   |     | E   |     |
| NDAMPM         |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| NEWSET         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| NINTPTS        | E   |     |     | E   |     |     | E   |     | E   |     | E   |     |     |
| NLAYERS        |   |     | E   |     |     |     |     |     |     |     |     | E   |     |
| NLHTLS         |   |     |     |     |     |     | E   |     |     |     |     | E   |     |
| NLMAX          |   |     |     |     |     |     |     |     |     |     |     | E   |     |

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| NLMIN          |   |     |     |     |     |     |     |     | E   | E   |     |     |     |
| NMLOOP         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| NOCOMPS        | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| NOELOF         | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| NOELOP         | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| NOGPF          | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| NOMSGSTR       | E   |     |     | E   |     |     | E   |     | E   |     |     |     |     |
| NONCUP         |   |     |     |     |     |     |     |     | B   |     |     |     |     |
| NQSET          | E   |     | E   | E   | E   | E   | E   |     | E   |     |     |     |     |
| NLTOL          |   |     |     |     |     |     |     |     | E   |     | B   |     |     |
| NUMOUT         | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| NUMOUT1        | E   |     |     | E   |     |     | E   |     | E   |     |     |     |     |

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| NUMOUT2        | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| OCMP           | E   |     |     |     |     |     |     |     |     | E   | E   |     |     |
| OEE            | E   |     |     |     |     |     |     |     |     | E   | E   | E   |     |
| OEF            | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OEFX           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OELMOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OELMSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OES            | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OESE           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OESX           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OG             | E   |     |     |     | E   |     |     | E   |     | E   | E   | E   |     |
| OGEM           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OGPF           | E   |     |     |     | E   |     |     |     |     | E   | E   | E   |     |
| OGPS           | E   |     |     |     | E   |     |     |     |     | E   |     |     |     |
| OGRDOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OGRDSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| OLDSEQ         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| OMAXR          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OMID           | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OMSGLVL        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OPCHSET        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OPG            |   |     | E   | E   |     | E   | E   |     | E   | E   | E   | E   | E   |
| OPGEOM         |   |     |     |     | B   | B   | B   |     |     | B   |     |     |     |
| OPGPKG         |   |     |     |     | B   | B   | B   |     |     | B   |     |     |     |
| OPPHIB         |   |     |     |     |     | B   | B   |     |     | B   |     |     |     |
| OPPHIPA        |   |     |     |     |     | B   | B   |     |     | B   |     |     |     |
| OPTEXIT        |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| OPTION         | B   |     |     |     |     |     |     |     |     |     |     |     |     |
| OQG            | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OSETELE        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OSETGRD        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OSWELM         | B   | B   | B   | B   | B   | B   | B   |     |     | B   | B   | B   |     |
| OSWPPT         | B   | B   | B   | B   | B   | B   | B   |     |     | B   | B   | B   |     |
| OUG            | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |
| OUGCORD        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OUGSPLIT       |   |     | E   | E   |     |     | E   |     | E   | E   | E   | E   |     |
| OUMU           | E   |     |     |     |     |     |     |     |     | E   |     |     |     |
| OUNIT1         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OUNIT2         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| OUTOPT         | E   |     |     | E   |     |     |     | E   |     | E   | E   | E   |     |
| PARTMEM        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| PATPLUS        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| PATVER         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| PDRMSG         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| PEDGEP         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| PENFN          |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| PH2OUT         |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| PLTMSG         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| POST           | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E*  |

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| POSTEXT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| POSTU          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PREFDB         |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| PRINT          |   |     |     |     |     | B   |     |     |     |     | B   |     |     |
| PROUT          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRPA           |   |     |     |     | E   |     |     | E   |     |     | E   |     |     |
| PRPHIVZ        | E   |     | E   | E   | E   | E   | E   |     | E   | E   |     |     |     |
| PRPJ           |   |     | E   |     | E   |     | E   | E   | E   | E   |     |     |     |
| PRTCSTM        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTEQXIN       | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTGPDT        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTGPL         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTGPTT        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTMAXIM       | E   |     | E   | E   | E   | E   | E   | E   | E   | E   |     |     |     |
| PRTMGG         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PRTPG          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| PSENPCH        |   |     |     |     |     |     |     |     |     | B   |     |     |     |

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| PVALINIT       |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| Q              |   |     |     |     |     |     | B   |     |     |     |     |     |     |
| RADMOD         |   |     |     |     |     |     |     | B   | B   |     | B   |     |     |
| RBSETPRT       | E   | E   | E   |     |     |     |     |     |     |     |     |     |     |
| RCONTACT       |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| RESLOPT        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| RHOCP          |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| RKSCHEME       |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| RMSINT         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| ROHYDRO        |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| ROMULTI        |   |     |     |     |     |     |     |     |     |     |     |     | B   |

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| ROSTR          |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| RSPECTRA       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| RSPRINT        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| RSTTEMP        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| S1             | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| S1A            | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| S1AG           | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| S1AM           | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| S1G            | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| S1M            | E   |     |     |     | E   |     |     | E   |     | E   | E   |     |     |
| SCRSPEC        | B   |     |     |     |     |     |     |     |     |     |     |     |     |
| SEMAP          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SEMAPOPT       | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SEMAPPRT       | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SENSUOO        |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| SEP1XOVR       | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SEQOUT         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SERST          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SESDAMP        | E   | E   | E   | E   |     | E   |     |     |     |     |     |     |     |
| SESEF          | E   |     |     |     |     |     |     |     |     |     |     |     |     |
| SHLDAMP        |   |     |     |     |     |     |     |     |     |     |     |     | E   |
| SIGMA          |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| SKINOUT        |   |     |     |     |     |     |     |     |     |     |     |     | E   |
| SKPAMP         |   |     |     |     |     | B   | B   |     |     |     |     |     | B   |
| SLOOPID        |   |     |     | E   |     |     |     |     |     |     |     |     |     |
| SMALLQ         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SNORM          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SNORMPRT       | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SOFTEXIT       |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| SOLADJC        |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| SOLID          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| SPARSEDM       |   |     |     |     | E   |     |     |     |     |     |     |     | E   |
| SPARSEDRL      |   |     |     |     | E   |     |     |     |     |     |     |     | E   |

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| SPARSEPH       |   |     |     |     |     | E   | E   |     |     | E   |     |     |     |
| SPDMRAT        |   |     |     |     |     |     |     |     |     | E   |     |     |     |
| SPDRRAT        |   |     |     |     | E   |     |     |     |     | E   |     |     |     |
| SQSETID        | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |
| STEPFCT        |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| STEPFCTL       |   |     |     |     |     |     |     |     |     |     |     |     | B   |
| SRTELTYP       | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| SRTOPT         | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| START          | B   |     | B   |     | B   | B   | B   | B   | B   | B   | B   | B   |     |
| STIME          |   |     |     | E   |     |     |     |     | E   |     |     |     |     |
| STRUCTMP       |   |     |     |     |     |     |     |     |     | E   |     |     |     |
| SUBCASID       |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| SUBID          |   |     |     |     |     |     |     | E   |     |     |     | E   |     |
| SUPAERO        |   |     |     |     | E   | E   | E   |     |     |     | E   |     |     |
| SUPDOF         |   |     |     |     | B   |     |     |     |     |     |     |     |     |
| SUPER          | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |
| TABID          |   |     |     |     |     |     |     |     |     |     |     |     |     |
| TABS           |   |     |     |     |     |     |     |     | E   | E   | E   |     |     |
| TESTNEG        |   |     |     |     |     |     |     |     | E   |     |     | E   |     |
| TFSYMFAC       |   |     | E   | E   |     | E   | E   | E   | E   | E   | E   | E   |     |

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| TINY           | E   |     |     |     | E   |     |     | E   |     | E   |     |     |     |
| TOLRSC         | B   |     | B   | B   | B   | B   | B   | B   | B   | B   | B   |     |     |
| TOPOCONV       |   |     |     |     |     |     |     |     |     |     |     | E   |     |
| TSTATIC        |   |     |     | E   |     |     |     |     | E   |     | E   | E   |     |
| UNSYMFM        |   |     |     |     |     |     |     |     |     |     | B   |     |     |
| UNSYMKB        | B   |     | B   | B   |     | B   | B   |     |     | B   | B   |     |     |
| UPDTBSH        |   |     |     |     |     |     |     |     |     | E   |     |     |     |
| USETPRT        | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11 (continued)

| PARAMeter Name | Solution Sequence Numbers (115 through 700) |     |     |     |     |     |     |     |     |     |     |     |     |
|----------------|---|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
|                | 115   | 116 | 118 | 129 | 144 | 145 | 146 | 153 | 159 | 200 | 400 | 600 | 700 |
| VARPHI         |   |     |     |     |     |     |     |     |     | C   |     |     |     |
| VELCUT         |   |     |     |     |     |     |     |     |     |     |     | B   |     |
| VMOPT          | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   |     |     |
| VREF           |   |     |     |     |     | B   |     |     |     | B   |     |     |     |
| VUBEAM         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUELJUMP       |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUGJUMP        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUHEXA         |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUPENTA        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUQUAD4        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| VUTETRA        |   |     |     |     |     |     |     |     |     |     |     |     |     |
| WTMASS         | E   |     | E   | E   | E   | E   | E   | E   | E   | E   | E   | B   |     |
| W3             |   |     |     | B   |     |     |     |     | E   | B   | E   | B   |     |
| W3FL           |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| W4             |   |     |     | B   |     |     |     |     | E   | B   | E   | B   |     |
| W4FL           |   |     |     |     |     |     |     |     |     | B   |     |     |     |
| XFLAG          |   |     |     |     | E   |     |     |     |     |     | E   |     |     |
| XYUNIT         |   |     |     |     | E   |     |     |     |     |     | E   |     |     |
| ZROCMAS        | E   |     | E   | E   | E   | E   | E   |     |     | E   | E   |     |     |
| ZROVEC         | E   |     | E   | E   | E   | E   | E   |     |     | E   | E   |     |     |

# 7

## Item Codes

- Item Code Description
- Element Stress (or Strain) Item Codes
- Element Force Item Codes
- Fluid Virtual Mass Pressure Item Codes
- 2D Slideline and 3D Surface Contact Item Codes
- Element Strain Energy Item Codes
- Fatigue Item Codes for LOC=ELEM on FTGPARM
- Fatigue Item Codes for LOC=NODE or NODA on FTGPARM
- Item Codes for Fatigue Analysis of Spot Welds
- Item Codes for Fatigue Analysis of Seam Welds
- Equivalent Radiated Power (ERP) Item Codes

## Item Code Description

Item codes are integer numbers assigned to specific output quantities; such as, the third translational component of displacement, the major principal stress at Z1 in a CQUAD4 element, or the torque in a CBAR element. Item codes are specified on the following input statements:

- DRESP1 entry for Design Sensitivity and Optimization (SOL 200).
- X-Y Plotting commands. See [Plotting](#) in the *MSC Nastran Reference Guide*.
- DTI,INDTA entry for stress sorting.

The following tables provide item codes for:

- [Table 7-1](#). Element Stress or Strain.
- [Table 7-6](#). Element Force.
- [Table 7-8](#). Fluid Virtual Mass Pressure.
- [Table 7-9](#). Heat Transfer Flux.
- [Table 7-10](#). Slideline Contact Output.
- [Table 7-13](#). Element Strain Energy Item Codes.
- [Table 7-14](#). Fatigue Item Codes (LOC=ELEM)
- [Table 7-15](#) Fatigue Item Codes (LOC=NODE or NODA)
- [Table 7-16](#). Item Codes for Fatigue Analysis of Spot Welds
- [Table 7-17](#). Item Codes for Fatigue Analysis of Seam Welds
- [Table 7-18](#) Equivalent Radiated Power (ERP) Item Codes

The following superscripts appear in the tables and indicate:

1. Data for components marked with the symbol (1) are included in the data block MES and MEF outputs from modules DRMH1/DRMH3 and DRMS1. DRMH1/DRMH3 are used in EXTSEOUT Case Control processing. (See [MSC Nastran DMAP Programmer's Guide](#).)
2. Composite Element Stresses and Failure Indices.

## Element Stress (or Strain) Item Codes

All item codes refer to stresses (or strains) unless otherwise denoted.

If output is magnitude/phase, the magnitude replaces the real part, and the phase replaces the imaginary part. Strain item codes are equivalent to stress item codes. However, strain is computed for only some elements. See [Table 3-1](#) in the *MSC Nastran Reference Guide*.

Table 7-1 Element Stress-Strain Item Codes

| Element Name (Code) | Real Stresses or Strains |                          | Complex Stresses or Strains |                          |                          |
|---------------------|--------------------------|--------------------------|-----------------------------|--------------------------|--------------------------|
|                     | Item Code                | Item                     | Item Code                   | Item                     | Real/Mag. or Imag./Phase |
| CAXIF2 (47)         | 2 <sup>1</sup>           | Radial axis              | 2 <sup>1</sup>              | Radial axis              | RM                       |
|                     | 3 <sup>1</sup>           | Axial axis               | 3 <sup>1</sup>              | Axial axis               | RM                       |
|                     | 4 <sup>1</sup>           | Tangential edge          | 4 <sup>1</sup>              | Tangential edge          | RM                       |
|                     | 5 <sup>1</sup>           | Circumferential edge     | 5 <sup>1</sup>              | Circumferential edge     | RM                       |
|                     |                          |                          | 6 <sup>1</sup>              | Radial axis              | IP                       |
|                     |                          |                          | 7 <sup>1</sup>              | Axial axis               | IP                       |
|                     |                          |                          | 8 <sup>1</sup>              | Tangential edge          | IP                       |
|                     |                          |                          | 9 <sup>1</sup>              | Circumferential edge     | IP                       |
|                     |                          |                          |                             |                          |                          |
| CAXIF3 (48)         | 2 <sup>1</sup>           | Radial centroid          | 2 <sup>1</sup>              | Radial centroid          | RM                       |
|                     | 3 <sup>1</sup>           | Circumferential centroid | 3 <sup>1</sup>              | Circumferential centroid | RM                       |
|                     | 4 <sup>1</sup>           | Axial centroid           | 4 <sup>1</sup>              | Axial centroid           | RM                       |
|                     | 5 <sup>1</sup>           | Tangential edge 1        | 5 <sup>1</sup>              | Tangential edge 1        | RM                       |
|                     | 6 <sup>1</sup>           | Circumferential edge 1   | 6 <sup>1</sup>              | Circumferential edge 1   | RM                       |
|                     | 7 <sup>1</sup>           | Tangential edge 2        | 7 <sup>1</sup>              | Tangential edge 2        | RM                       |
|                     | 8 <sup>1</sup>           | Circumferential edge 2   | 8 <sup>1</sup>              | Circumferential edge 2   | RM                       |
|                     | 9 <sup>1</sup>           | Tangential edge 3        | 9 <sup>1</sup>              | Tangential edge 3        | RM                       |
|                     | 10 <sup>1</sup>          | Circumferential edge 3   | 10 <sup>1</sup>             | Circumferential edge 3   | RM                       |
|                     |                          |                          | 11 <sup>1</sup>             | Radial centroid          | IP                       |
|                     |                          |                          | 12 <sup>1</sup>             | Circumferential centroid | IP                       |
|                     |                          |                          | 13 <sup>1</sup>             | Axial centroid           | IP                       |
|                     |                          |                          | 14 <sup>1</sup>             | Tangential edge 1        | IP                       |
|                     |                          |                          | 15 <sup>1</sup>             | Circumferential edge 1   | IP                       |
|                     |                          |                          | 16 <sup>1</sup>             | Tangential edge 2        | IP                       |
|                     |                          |                          | 17 <sup>1</sup>             | Circumferential edge 2   | IP                       |
|                     |                          |                          | 18 <sup>1</sup>             | Tangential edge 3        | IP                       |
|                     |                          |                          | 19 <sup>1</sup>             | Circumferential edge 3   | IP                       |
| CAXIF4 (49)         | 2 <sup>1</sup>           | Radial centroid          | 2 <sup>1</sup>              | Radial centroid          | RM                       |
|                     | 3 <sup>1</sup>           | Circumferential centroid | 3 <sup>1</sup>              | Circumferential centroid | RM                       |
|                     | 4 <sup>1</sup>           | Axial centroid           | 4 <sup>1</sup>              | Axial centroid           | RM                       |
|                     | 5 <sup>1</sup>           | Tangential edge 1        | 5 <sup>1</sup>              | Tangential edge 1        | RM                       |
|                     | 6 <sup>1</sup>           | Circumferential edge 1   | 6 <sup>1</sup>              | Circumferential edge 1   | RM                       |

Table 7-1 Element Stress-Strain Item Codes (continued)

| Element Name (Code) | Real Stresses or Strains |  | Complex Stresses or Strains |                          |                          |
|---------------------|--------------------------|--|-----------------------------|--------------------------|--------------------------|
|                     | Item Code                | Item   | Item Code                   | Item                     | Real/Mag. or Imag./Phase |
|                     | 7 <sup>1</sup>           | Tangential edge 2                                | 7 <sup>1</sup>              | Tangential edge 2        | RM                       |
|                     | 8 <sup>1</sup>           | Circumferential edge 2                           | 8 <sup>1</sup>              | Circumferential edge 2   | RM                       |
|                     | 9 <sup>1</sup>           | Tangential edge 3                                | 9 <sup>1</sup>              | Tangential edge 3        | RM                       |
|                     | 10 <sup>1</sup>          | Circumferential edge 3                           | 10 <sup>1</sup>             | Circumferential edge 3   | RM                       |
|                     | 11 <sup>1</sup>          | Tangential edge 4                                | 11 <sup>1</sup>             | Tangential edge 4        | RM                       |
|                     | 12 <sup>1</sup>          | Circumferential edge 4                           | 12 <sup>1</sup>             | Circumferential edge 4   | RM                       |
|                     |                          |  | 13                          | Radial centroid          | IP                       |
|                     |                          |  | 14                          | Circumferential centroid | IP                       |
|                     |                          |  | 15                          | Axialcentroid            | IP                       |
|                     |                          |  | 16                          | Tangential edge 1        | IP                       |
|                     |                          |  | 17                          | Circumferential edge 1   | IP                       |
|                     |                          |  | 18                          | Tangential edge 2        | IP                       |
|                     |                          |  | 19                          | Circumferential edge 2   | IP                       |
|                     |                          |  | 20                          | Tangential edge 3        | IP                       |
|                     |                          |  | 21                          | Circumferential edge 3   | IP                       |
|                     |                          |  | 22                          | Tangential edge 4        | IP                       |
|                     |                          |  | 23                          | Circumferential edge 4   | IP                       |
| CAXISYM (241)       | 2                        | Z1-Fiber distance 1                              |                             |                          |                          |
| Nonlinear           | 3                        | Stress-X   |                             |                          |                          |
|                     | 4                        | Stress-Y   |                             |                          |                          |
|                     | 5                        | Stress-XY  |                             |                          |                          |
|                     | 6                        | Strain-X   |                             |                          |                          |
|                     | 7                        | Strain-Y   |                             |                          |                          |
|                     | 8                        | Strain XY  |                             |                          |                          |
|                     | 9-15                     | Item 9 through 15 repeated for fiber distance Z2 |                             |                          |                          |
| CBAR (34)           | 2 <sup>1</sup>           | End A-Point C                                    | 2 <sup>1</sup>              | End A-Point C            | RM                       |
|                     | 3 <sup>1</sup>           | End A-Point D                                    | 3 <sup>1</sup>              | End A-Point D            | RM                       |
|                     | 4 <sup>1</sup>           | End A-Point E                                    | 4 <sup>1</sup>              | End A-Point E            | RM                       |
|                     | 5 <sup>1</sup>           | End A-Point F                                    | 5 <sup>1</sup>              | End A-Point F            | RM                       |
|                     | 6 <sup>1</sup>           | Axial  | 6 <sup>1</sup>              | Axial                    | RM                       |
|                     | 7                        | End A maximum                                    | 7 <sup>1</sup>              | End A-Point C            | IP                       |
|                     | 8                        | End A minimum                                    | 8 <sup>1</sup>              | End A-Point D            | IP                       |

**Table 7-1 Element Stress-Strain Item Codes (continued)**

| Element Name (Code)   | Real Stresses or Strains |   | Complex Stresses or Strains |  |                          |
|-----------------------|--------------------------|---|-----------------------------|--|--------------------------|
|                       | Item Code                | Item  | Item Code                   | Item   | Real/Mag. or Imag./Phase |
|                       | 9                        | Safety margin in tension  | 9 <sup>1</sup>              | End A-Point E  | IP                       |
|                       | 10 <sup>1</sup>          | End B-Point C   | 10 <sup>1</sup>             | End A-Point F  | IP                       |
|                       | 11 <sup>1</sup>          | End B-Point D   | 11 <sup>1</sup>             | Axial  | IP                       |
|                       | 12 <sup>1</sup>          | End B-Point E   | 12 <sup>1</sup>             | End B-Point C  | RM                       |
|                       | 13 <sup>1</sup>          | End B-Point F   | 13 <sup>1</sup>             | End B-Point D  | RM                       |
|                       | 14                       | End B maximum   | 14 <sup>1</sup>             | End B-Point E  | RM                       |
|                       | 15                       | End B minimum   | 15 <sup>1</sup>             | End B-Point F  | RM                       |
|                       | 16                       | Safety margin in compression  | 16 <sup>1</sup>             | End B-Point C  | IP                       |
|                       |                          |   | 17 <sup>1</sup>             | End B-Point D  | IP                       |
|                       |                          |   | 18 <sup>1</sup>             | End B-Point E  | IP                       |
|                       |                          |   | 19 <sup>1</sup>             | End B-Point F  | IP                       |
| <b>CBAR (100)</b>     | 2                        | Station Distance/Length   | 2                           | Station Distance/Length  | RM                       |
| Intermediate Stations | 3                        | Point C   | 3                           | Point C  | RM                       |
|                       | 4                        | Point D   | 4                           | Point D  | RM                       |
|                       | 5                        | Point E   | 5                           | Point E  | RM                       |
|                       | 6                        | Point F   | 6                           | Point F  | RM                       |
|                       | 7                        | Axial   | 7                           | Axial  | RM                       |
|                       | 8                        | Maximum   | 8                           | Maximum  | RM                       |
|                       | 9                        | Minimum   | 9                           | Minimum  | RM                       |
|                       | 10                       | Margin of Safety  | 10                          | Point C  | IP                       |
|                       |                          | (Item codes above are given for End A. For codes 2 through 10 at intermediate stations add $(K-1)*9$ where K is the station number, and for codes at End B, K=number of stations plus 1.) | 11                          | Point D  | IP                       |
|                       |                          |   | 12                          | Point E  | IP                       |
|                       |                          |   | 13                          | Point F  | IP                       |
|                       |                          |   | 14                          | Axial  | IP                       |
|                       |                          |   | 15                          | Maximum  | IP                       |
|                       |                          |   | 16                          | Minimum  |                          |
|                       |                          |   |                             | (Item codes above are given for End A. For codes 2 through 16 at intermediate stations add $(K-1)*15$ where K is the station number, and for codes at End B, K=number of stations plus 1.) |                          |

Table 7-1 Element Stress-Strain Item Codes (continued)

| Element Name (Code)                          | Real Stresses or Strains |  | Complex Stresses or Strains |  |                          |
|--|--------------------------|--|-----------------------------|--|--------------------------|
|  | Item Code                | Item   | Item Code                   | Item   | Real/Mag. or Imag./Phase |
| <b>CBAR (238)</b><br>Arbitrary Cross Section | 2                        | Station Distance/Length  |                             |  |                          |
|  | 3                        | Max Axial Stress   |                             |  |                          |
|  | 4                        | Min Axial Stress   |                             |  |                          |
|  | 5                        | Max Shear Stress in xy   |                             |  |                          |
|  | 6                        | Min Shear Stress in xy   |                             |  |                          |
|  | 7                        | Max Shear Stress in zx   |                             |  |                          |
|  | 8                        | Min Shear Stress in zx   |                             |  |                          |
|  | 9                        | Max vonMises Stress  |                             | (Not Supported)  |                          |
|  |                          |  |                             |  |                          |
| <b>CBEAM (2)</b><br>Linear                   | 2                        | External grid point ID   | 2                           | External grid point ID   |                          |
|  | 3                        | Station distance/length  | 3                           | Station distance/length  |                          |
|  | 4 <sup>1</sup>           | Long. Stress at Point C  | 4 <sup>1</sup>              | Long. Stress at Point C  | RM                       |
|  | 5 <sup>1</sup>           | Long. Stress at Point D  | 5 <sup>1</sup>              | Long. Stress at Point D  | RM                       |
|  | 6 <sup>1</sup>           | Long. Stress at Point E  | 6 <sup>1</sup>              | Long. Stress at Point E  | RM                       |
|  | 7 <sup>1</sup>           | Long. Stress at Point F  | 7 <sup>1</sup>              | Long. Stress at Point F  | RM                       |
|  | 8                        | Maximum stress   | 8 <sup>1</sup>              | Long. Stress at Point C  | IP                       |
|  | 9                        | Minimum stress   | 9 <sup>1</sup>              | Long. Stress at Point D  | IP                       |
|  | 10                       | Safety margin in tension   | 10 <sup>1</sup>             | Long. Stress at Point E  | IP                       |
|  | 11                       | Safety margin in compression   | 11 <sup>1</sup>             | Long. Stress at Point F  | IP                       |
|  |                          | (Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.) |                             | (Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.) |                          |
|  |                          |  |                             |  |                          |
| <b>CBEAM (94)</b><br>Nonlinear               | 2                        | External grid point ID   |                             | Not applicable   |                          |
|  | 3                        | C (Character)  |                             |  |                          |
|  | 4                        | Long. Stress at point C  |                             |  |                          |
|  | 5                        | Equivalent stress  |                             |  |                          |
|  | 6                        | Total strain   |                             |  |                          |
|  | 7                        | Effective plastic strain   |                             |  |                          |
|  | 8                        | Effective creep strain   |                             |  |                          |
|  |                          | (Item codes 3 through 8 are repeated for points D, E, and F. Then the entire record (from 2 through N) is repeated for end B of the element.)  |                             |  |                          |

Table 7-1 Element Stress-Strain Item Codes (continued)

| Element Name (Code)                           | Real Stresses or Strains |  | Complex Stresses or Strains |  |                          |
|---|--------------------------|--|-----------------------------|--|--------------------------|
|   | Item Code                | Item                                   | Item Code                   | Item                                   | Real/Mag. or Imag./Phase |
| <b>CBEAM (239)</b><br>Arbitrary Cross Section | 2                        | Station Distance/Length                |                             |  |                          |
|   | 3                        | Max Axial Stress                       |                             |  |                          |
|   | 4                        | Min Axial Stress                       |                             |  |                          |
|   | 5                        | Max Shear Stress in xy                 |                             |  |                          |
|   | 6                        | Min Shear Stress in xy                 |                             |  |                          |
|   | 7                        | Max Shear Stress in xz                 |                             |  |                          |
|   | 8                        | Min Shear Stress in xz                 |                             |  |                          |
|   | 9                        | Max von Mises Stress                   |                             | (Not supported)                        |                          |
|   | 2                        | External grid point ID                 | 2                           | External grid point ID                 |                          |
|   | 3                        | Normal stress at point C               | 3                           | Normal stress at point C               | RM                       |
| <b>CBEAM3 (184)</b><br>Linear                 | 4                        | Normal stress at point D               | 4                           | Normal stress at point D               | RM                       |
|   | 5                        | Normal stress at point E               | 5                           | Normal stress at point E               | RM                       |
|   | 6                        | Normal stress at point F               | 6                           | Normal stress at point F               | RM                       |
|   | 7                        | Maximum stress                         | 7                           | Normal stress at point C               | IP                       |
|   | 8                        | Minimum stress                         | 8                           | Normal stress at point D               | IP                       |
|   | 9                        | Safety margin in tension               | 9                           | Normal stress at point E               | IP                       |
|   | 10                       | Safety margin in compression           | 10                          | Normal stress at point F               | IP                       |
|   | 11                       | Shear stress in y-direction at point C | 11                          | Shear stress in y-direction at point C | RM                       |
|   | 12                       | Shear stress in y-direction at point D | 12                          | Shear stress in y-direction at point D | RM                       |
|   | 13                       | Shear stress in y-direction at point E | 13                          | Shear stress in y-direction at point E | RM                       |
|   | 14                       | Shear stress in y-direction at point F | 14                          | Shear stress in y-direction at point F | RM                       |
|   | 15                       | Maximum shear stress in y-direction    | 15                          | Shear stress in y-direction at point C | IP                       |
|   | 16                       | Minimum shear stress in y-direction    | 16                          | Shear stress in y-direction at point D | IP                       |
|   | 17                       | (Not applicable)                       | 17                          | Shear stress in y-direction at point E | IP                       |
|   | 18                       | (Not applicable)                       | 18                          | Shear stress in y-direction at point F | IP                       |
|   | 19                       | Shear stress in z-direction at point C | 19                          | Shear stress in z-direction at point C | RM                       |

Table 7-1 Element Stress-Strain Item Codes (continued)

| Element Name (Code) | Real Stresses or Strains |   | Complex Stresses or Strains |  |                          |
|---------------------|--------------------------|---|-----------------------------|--|--------------------------|
|                     | Item Code                | Item  | Item Code                   | Item   | Real/Mag. or Imag./Phase |
|                     | 20                       | Shear stress in z-direction at point D  | 20                          | Shear stress in z-direction at point D   | RM                       |
|                     | 21                       | Shear stress in z-direction at point E  | 21                          | Shear stress in z-direction at point E   | RM                       |
|                     | 22                       | Shear stress in z-direction at point F  | 22                          | Shear stress in z-direction at point F   | RM                       |
|                     | 23                       | Maximum shear stress in z-direction   | 23                          | Shear stress in z-direction at point C   | IP                       |
|                     | 24                       | Minimum shear stress in z-direction   | 24                          | Shear stress in z-direction at point D   | IP                       |
|                     | 25                       | (Not applicable)  | 25                          | Shear stress in z-direction at point Z   | IP                       |
|                     | 26                       | (Not applicable)<br><br>(Item codes are given for end A. they are repeated for end B and mid-node C, respectively)                      | 26                          | Shear stress in z-direction at point F<br><br>(Item codes are given for end A. They are repeated for end B and mid-node C, respectively) | IP                       |
| <b>CBEND (69)</b>   | 2                        | External grid point ID  | 2                           | External grid point ID   |                          |
|                     | 3                        | Circumferential angle   | 3                           | Circumferential angle  |                          |
|                     | 4 <sup>1</sup>           | Long. Stress at Point C   | 4 <sup>1</sup>              | Long. Stress at Point C  | RM                       |
|                     | 5 <sup>1</sup>           | Long. Stress at Point D   | 5 <sup>1</sup>              | Long. Stress at Point D  | RM                       |
|                     | 6 <sup>1</sup>           | Long. Stress at Point E   | 6 <sup>1</sup>              | Long. Stress at Point E  | RM                       |
|                     | 7 <sup>1</sup>           | Long. Stress at Point F   | 7 <sup>1</sup>              | Long. Stress at Point F  | RM                       |
|                     | 8                        | Maximum stress  | 8 <sup>1</sup>              | Long. Stress at Point C  | IP                       |
|                     | 9                        | Minimum stress  | 9 <sup>1</sup>              | Long. Stress at Point D  | IP                       |
|                     | 10                       | Safety margin in tension  | 10 <sup>1</sup>             | Long. Stress at Point E  | IP                       |
|                     | 11                       | Safety margin in compression<br><br>(Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.) | 11 <sup>1</sup>             | Long. Stress at Point F<br><br>(Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)       | IP                       |
| <b>CBUSH (102)</b>  | 2 <sup>1</sup>           | Translation-x   | 2 <sup>1</sup>              | Translation-x  | R                        |
|                     | 3 <sup>1</sup>           | Translation-y   | 3 <sup>1</sup>              | Translation-y  | R                        |
|                     | 4 <sup>1</sup>           | Translation-z   | 4 <sup>1</sup>              | Translation-z  | R                        |
|                     | 5 <sup>1</sup>           | Rotation-x  | 5 <sup>1</sup>              | Rotation-x   | R                        |
|                     | 6 <sup>1</sup>           | Rotation-y  | 6 <sup>1</sup>              | Rotation-y   | R                        |
|                     | 7 <sup>1</sup>           | Rotation-z  | 7 <sup>1</sup>              | Rotation-z   | R                        |
|                     |                          |   | 8 <sup>1</sup>              | Translation-x  | I                        |

Table 7-1 Element Stress-Strain Item Codes (continued)

| Element Name (Code) | Real Stresses or Strains |                    | Complex Stresses or Strains |                | Real/Mag. or Imag./Phase |
|---------------------|--------------------------|--------------------|-----------------------------|----------------|--------------------------|
|                     | Item Code                | Item               | Item Code                   | Item           |                          |
|                     |                          |                    | 9 <sup>1</sup>              | Translation-y  | I                        |
|                     |                          |                    | 10 <sup>1</sup>             | Translation-z  | I                        |
|                     |                          |                    | 11 <sup>1</sup>             | Rotation-x     | I                        |
|                     |                          |                    | 12 <sup>1</sup>             | Rotation-y     | I                        |
|                     |                          |                    | 13 <sup>1</sup>             | Rotation-z     | I                        |
| CBUSH1D (40)        | 1                        | Element ID         |                             |                |                          |
|                     | 2                        | Axial force        |                             |                |                          |
|                     | 3                        | Axial displacement |                             |                |                          |
|                     | 4                        | Axial velocity     |                             | Not applicable |                          |
|                     | 5                        | Axial stress       |                             |                |                          |
|                     | 6                        | Axial strain       |                             |                |                          |
|                     | 7                        |                    |                             |                |                          |
|                     | 8                        |                    |                             |                |                          |

Table 7-2 Element Stress-Strain Item Codes Part 2

| Element Name (Code) | Real Stresses or Strains |                         | Complex Stresses or Strains |                | Real/Mag. or Imag./Phase |
|---------------------|--------------------------|-------------------------|-----------------------------|----------------|--------------------------|
|                     | Item Code                | Item                    | Item Code                   | Item           |                          |
| CCONEAX (35)        | 2                        | Harmonic or point angle |                             | Not applicable |                          |
|                     | 3                        | Z1=Fiber Distance 1     |                             |                |                          |
|                     | 4 <sup>1</sup>           | Normal v at Z1          |                             |                |                          |
|                     | 5 <sup>1</sup>           | Normal u at Z1          |                             |                |                          |
|                     | 6 <sup>1</sup>           | Shear uv at Z1          |                             |                |                          |
|                     | 7                        | Shear angle at Z1       |                             |                |                          |
|                     | 8                        | Major principal at Z1   |                             |                |                          |
|                     | 9                        | Minor principal at Z1   |                             |                |                          |
|                     | 10                       | Maximum shear at Z1     |                             |                |                          |
|                     | 11                       | Z2= Fiber Distance 2    |                             |                |                          |
|                     | 12 <sup>1</sup>          | Normal v at Z2          |                             |                |                          |
|                     | 13 <sup>1</sup>          | Normal u at Z2          |                             |                |                          |
|                     | 14 <sup>1</sup>          | Shear uv at Z2          |                             |                |                          |
|                     | 15                       | Shear angle at Z2       |                             |                |                          |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code) | Real Stresses or Strains |                             | Complex Stresses or Strains |                             |                             |
|------------------------|--------------------------|-----------------------------|-----------------------------|-----------------------------|-----------------------------|
|                        | Item<br>Code             | Item                        | Item<br>Code                | Item                        | Real/Mag. or<br>Imag./Phase |
|                        | 16                       | Major principal at Z2       |                             |                             |                             |
|                        | 17                       | Minor principal at Z2       |                             |                             |                             |
|                        | 18                       | Maximum shear at Z2         |                             |                             |                             |
| CDUM3                  | 2 <sup>1</sup>           | S1                          | 2 <sup>1</sup>              | S1                          | RM                          |
| thru                   | 3 <sup>1</sup>           | S2                          | 3 <sup>1</sup>              | S2                          | RM                          |
| CDUM9 (55-61)          | 4 <sup>1</sup>           | S3                          | 4 <sup>1</sup>              | S3                          | RM                          |
|                        | 5 <sup>1</sup>           | S4                          | 5 <sup>1</sup>              | S4                          | RM                          |
|                        | 6 <sup>1</sup>           | S5                          | 6 <sup>1</sup>              | S5                          | RM                          |
|                        | 7 <sup>1</sup>           | S6                          | 7 <sup>1</sup>              | S6                          | RM                          |
|                        | 8 <sup>1</sup>           | S7                          | 8 <sup>1</sup>              | S7                          | RM                          |
| CELAS1 (11)            | 2 <sup>1</sup>           | Stress                      | 2 <sup>1</sup>              | Stress                      | RM                          |
|                        |                          |                             | 3 <sup>1</sup>              | Stress                      | IP                          |
| CELAS2 (12)            | 2 <sup>1</sup>           | Stress                      | 2 <sup>1</sup>              | Stress                      | RM                          |
|                        |                          |                             | 3 <sup>1</sup>              | Stress                      | IP                          |
| CELAS3 (13)            | 2 <sup>1</sup>           | Stress                      | 2 <sup>1</sup>              | Stress                      | RM                          |
|                        |                          |                             | 3 <sup>1</sup>              | Stress                      | IP                          |
| CGAP (86)              | 2                        | Normal x                    |                             | Not applicable              |                             |
|                        | 3                        | Shear y                     |                             |                             |                             |
|                        | 4                        | Shear z                     |                             |                             |                             |
|                        | 5                        | Axial u                     |                             |                             |                             |
|                        | 6                        | Shear v                     |                             |                             |                             |
|                        | 7                        | Shear w                     |                             |                             |                             |
|                        | 8                        | Slip v                      |                             |                             |                             |
|                        | 9                        | Slip w                      |                             |                             |                             |
| CHEXA (67)             | 2                        | Stress coordinate system    | 2                           | Stress coordinate system    |                             |
| Linear                 | 3                        | Coordinate type (Character) | 3                           | Coordinate type (Character) |                             |
|                        | 4                        | Number of active points     | 4                           | Number of active points     |                             |
|                        | 5                        | External grid ID (0=center) | 5                           | External grid ID (0=center) |                             |
|                        | 6 <sup>1</sup>           | Normal x                    | 6 <sup>1</sup>              | Normal x                    | RM                          |
|                        | 7 <sup>1</sup>           | Shear xy                    | 7 <sup>1</sup>              | Normal y                    | RM                          |
|                        | 8                        | First principal             | 8 <sup>1</sup>              | Normal z                    | RM                          |
|                        | 9                        | First principal x cosine    | 9 <sup>1</sup>              | Shear xy                    | RM                          |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |   | Complex Stresses or Strains |   |                                |
|-------------------------|--------------------------|---|-----------------------------|---|--------------------------------|
|                         | Item<br>Code             | Item                                      | Item<br>Code                | Item                                      | Real/Mag.<br>or<br>Imag./Phase |
|                         | 10                       | Second principal x cosine                 | 10 <sup>1</sup>             | Shear yz                                  | RM                             |
|                         | 11                       | Third principal x cosine                  | 11 <sup>1</sup>             | Shear zx                                  | RM                             |
|                         | 12                       | Mean pressure                             | 12 <sup>1</sup>             | Normal x                                  | IP                             |
|                         | 13                       | von Mises or octahedral shear stress      | 13 <sup>1</sup>             | Normal y                                  | IP                             |
|                         | 14 <sup>1</sup>          | Normal y                                  | 14 <sup>1</sup>             | Normal z                                  | IP                             |
|                         | 15 <sup>1</sup>          | Shear yz                                  | 15 <sup>1</sup>             | Shear xy                                  | IP                             |
|                         | 16                       | Second principal                          | 16 <sup>1</sup>             | Shear yz                                  | IP                             |
|                         | 17                       | First principal y cosine                  | 17 <sup>1</sup>             | Shear zx                                  | IP                             |
|                         | 18                       | Second principal y cosine                 | 18-121                      | Items 5 through 17 repeated for 8 corners |                                |
|                         | 19                       | Third principal y cosine                  |                             |   |                                |
|                         | 20 <sup>1</sup>          | Normal z                                  |                             |   |                                |
|                         | 21 <sup>1</sup>          | Shear zx                                  |                             |   |                                |
|                         | 22                       | Third principal                           |                             |   |                                |
|                         | 23                       | First principal z cosine                  |                             |   |                                |
|                         | 24                       | Second principal z cosine                 |                             |   |                                |
|                         | 25                       | Third principal z cosine                  |                             |   |                                |
|                         | 26-193                   | Items 5 through 25 repeated for 8 corners |                             |   |                                |
| CHEXA (93)<br>Nonlinear | 2                        | Stress coordinate system                  |                             |   |                                |
|                         | 3                        | Grid/Gauss                                |                             |   |                                |
|                         | 4                        | Number of active points                   |                             |   |                                |
|                         | 5                        | External grid ID (0=center)               |                             |   |                                |
|                         | 6                        | Stress-X                                  |                             |   |                                |
|                         | 7                        | Stress-Y                                  |                             |   |                                |
|                         | 8                        | Stress-Z                                  |                             |   |                                |
|                         | 9                        | Stress-XY                                 |                             |   |                                |
|                         | 10                       | Stress-YZ                                 |                             |   |                                |
|                         | 11                       | Stress-ZX                                 |                             |   |                                |
|                         | 12                       | Equivalent stress                         |                             |   |                                |
|                         | 13                       | Effective plastic strain                  |                             | Not applicable                            |                                |
|                         | 14                       | Effective creep strain                    |                             |   |                                |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |   | Complex Stresses or Strains |                |                             |
|--|--------------------------|---|-----------------------------|----------------|-----------------------------|
|  | Item<br>Code             | Item  | Item<br>Code                | Item           | Real/Mag. or<br>Imag./Phase |
|  | 15                       | Strain-X  |                             |                |                             |
|  | 16                       | Strain-Y  |                             |                |                             |
|  | 17                       | Strain-Z  |                             |                |                             |
|  | 18                       | Strain-XY   |                             |                |                             |
|  | 19                       | Strain-YZ   |                             |                |                             |
|  | 20                       | Strain-ZX   |                             |                |                             |
|  | 21-148                   | Items 5 through 20<br>Repeated for 8 corners                  |                             |                |                             |
| CHEXA FD (202)<br>Nonlinear Finite<br>Deformation with 8<br>grid points  | 2                        | Grid/Gauss  |                             |                |                             |
|  | 3                        | Grid/Gauss ID (0=center)                                      |                             |                |                             |
|  | 4                        | Cauchy stress-X   |                             |                |                             |
|  | 5                        | Cauchy stress-Y   |                             |                |                             |
|  | 6                        | Cauchy stress-Z   |                             |                |                             |
|  | 7                        | Cauchy stress-XY  |                             |                |                             |
|  | 8                        | Cauchy stress-YZ  |                             |                |                             |
|  | 9                        | Cauchy stress-ZX  |                             |                |                             |
|  | 10                       | Pressure<br>$p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$ |                             | Not applicable |                             |
|  | 11                       | Volumetric strain J-1   |                             |                |                             |
|  | 12                       | Logarithmic strain-X  |                             |                |                             |
|  | 13                       | Logarithmic strain-Y  |                             |                |                             |
|  | 14                       | Logarithmic strain-Z  |                             |                |                             |
|  | 15                       | Logarithmic strain-XY   |                             |                |                             |
|  | 16                       | Logarithmic strain-YZ   |                             |                |                             |
|  | 17                       | Logarithmic strain-ZX   |                             |                |                             |
|  | 18-122                   | Items 3 through 17 repeated<br>for 7 Gauss/grid points        |                             |                |                             |
| CHEXA FD (207)<br>Nonlinear Finite<br>Deformation with<br>20 grid points | 2-17                     | Same as CHEXA FD (202)  |                             |                |                             |
|  | 18-407                   | Items 3 through 17 repeated<br>for 26 Gauss points            |                             | Not applicable |                             |
| CIFHEX (65)<br>Nonlinear   | 2                        | Grid/Gauss  |                             |                |                             |
|  | 3                        | External Grid ID (0 =<br>Center)                              |                             |                |                             |
|  | 4                        | Normal Stress   |                             |                |                             |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)    | Real Stresses or Strains |   | Complex Stresses or Strains |      |                             |
|---------------------------|--------------------------|---|-----------------------------|------|-----------------------------|
|                           | Item<br>Code             | Item                                      | Item<br>Code                | Item | Real/Mag. or<br>Imag./Phase |
|                           | 5                        | Shear Stress 1                            |                             |      |                             |
|                           | 6                        | Shear Stress 2                            |                             |      |                             |
|                           | 7                        | Normal Strain                             |                             |      |                             |
|                           | 8                        | Shear Strain 1                            |                             |      |                             |
|                           | 9                        | Shear Strain 2                            |                             |      |                             |
|                           | 10                       | Damage Value                              |                             |      |                             |
|                           | 11-74                    | Items 3 through 10 repeated for 6 Corners |                             |      |                             |
| CIPENT (66)<br>Nonlinear  | 2                        | Grid/Gauss                                |                             |      |                             |
|                           | 3                        | External Grid ID (0 = Center)             |                             |      |                             |
|                           | 4                        | Normal Stress                             |                             |      |                             |
|                           | 5                        | Shear Stress 1                            |                             |      |                             |
|                           | 6                        | Shear Stress 2                            |                             |      |                             |
|                           | 7                        | Normal Strain                             |                             |      |                             |
|                           | 8                        | Shear Strain 1                            |                             |      |                             |
|                           | 9                        | Shear Strain 2                            |                             |      |                             |
|                           | 10                       | Damage Value                              |                             |      |                             |
|                           | 11-58                    | Items 3 through 10 repeated for 6 Corners |                             |      |                             |
| CIFQDX (73)<br>Nonlinear  | 2                        | Grid/Gauss                                |                             |      |                             |
|                           | 3                        | External Grid ID (0 = Center)             |                             |      |                             |
|                           | 4                        | Normal Stress                             |                             |      |                             |
|                           | 5                        | Shear Stress                              |                             |      |                             |
|                           | 6                        | Normal Strain                             |                             |      |                             |
|                           | 7                        | Shear Strain                              |                             |      |                             |
|                           | 8                        | Damage Value                              |                             |      |                             |
|                           | 9-32                     | Items 3 through 8 repeated for 4 Corners  |                             |      |                             |
| CIFQUAD (63)<br>Nonlinear | 2                        | Grid/Gauss                                |                             |      |                             |
|                           | 3                        | External Grid ID (0 = Center)             |                             |      |                             |
|                           | 4                        | Normal Stress                             |                             |      |                             |
|                           | 5                        | Shear Stress                              |                             |      |                             |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code) | Real Stresses or Strains |  | Complex Stresses or Strains |                             |                             |
|------------------------|--------------------------|--|-----------------------------|-----------------------------|-----------------------------|
|                        | Item<br>Code             | Item                                     | Item<br>Code                | Item                        | Real/Mag. or<br>Imag./Phase |
|                        | 6                        | Normal Strain                            |                             |                             |                             |
|                        | 7                        | Shear Strain                             |                             |                             |                             |
|                        | 8                        | Damage Value                             |                             |                             |                             |
|                        | 9-32                     | Items 3 through 8 repeated for 4 Corners |                             |                             |                             |
| <b>CONROD (10)</b>     | 2 <sup>1</sup>           | Axial stress                             | 2 <sup>1</sup>              | Axial stress                | RM                          |
| Linear                 | 3                        | Axial safety margin                      | 3 <sup>1</sup>              | Axial stress                | IP                          |
|                        | 4 <sup>1</sup>           | Torsional stress                         | 4 <sup>1</sup>              | Torsional stress            | RM                          |
|                        | 5                        | Torsional safety margin                  | 5 <sup>1</sup>              | Torsional stress            | IP                          |
| <b>CONROD (92)</b>     | 2                        | Axial stress                             |                             |                             |                             |
| Nonlinear              | 3                        | Equivalent stress                        |                             |                             |                             |
|                        | 4                        | Total strain                             |                             |                             |                             |
|                        | 5                        | Effective plastic strain                 |                             | Not applicable              |                             |
|                        | 6                        | Effective creep strain                   |                             |                             |                             |
|                        | 7                        | Linear torsional stress                  |                             |                             |                             |
| <b>CPENTA (68)</b>     | 2                        | Stress coordinate system                 | 2                           | Stress coordinate system    |                             |
| Linear                 | 3                        | Coordinate type (Character)              | 3                           | Coordinate type (Character) |                             |
|                        | 4                        | Number of active points                  | 4                           | Number of active points     |                             |
|                        | 5                        | External grid ID                         | 5                           | External grid ID            |                             |
|                        |                          | (0=center)                               |                             | (0=center)                  |                             |
|                        | 6 <sup>1</sup>           | Normal x                                 | 6 <sup>1</sup>              | Normal x                    | RM                          |
|                        | 7 <sup>1</sup>           | Shear xy                                 | 7 <sup>1</sup>              | Normal y                    | RM                          |
|                        | 8                        | First principal                          | 8 <sup>1</sup>              | Normal z                    | RM                          |
|                        | 9                        | First principal x cosine                 | 91                          | Shear xy                    | RM                          |
|                        | 10                       | Second principal x cosine                | 10 <sup>1</sup>             | Shear yz                    | RM                          |
|                        | 11                       | Third principal x cosine                 | 11 <sup>1</sup>             | Shear zx                    | RM                          |
|                        | 12                       | Mean pressure                            | 12 <sup>1</sup>             | Normal x                    | IP                          |
|                        | 13                       | von Mises or Octahedral shear stress     | 13 <sup>1</sup>             | Normal y                    | IP                          |
|                        | 14 <sup>1</sup>          | Normal y                                 | 14 <sup>1</sup>             | Normal z                    | IP                          |
|                        | 15 <sup>1</sup>          | Shear yz                                 | 15 <sup>1</sup>             | Shear xy                    | IP                          |
|                        | 16                       | Second principal                         | 16 <sup>1</sup>             | Shear yz                    | IP                          |
|                        | 17                       | First principal y cosine                 | 17 <sup>1</sup>             | Shear zx                    | IP                          |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)              | Real Stresses or Strains |  | Complex Stresses or Strains |   |                             |
|-------------------------------------|--------------------------|--|-----------------------------|---|-----------------------------|
|                                     | Item<br>Code             | Item   | Item<br>Code                | Item                                      | Real/Mag. or<br>Imag./Phase |
|                                     | 18                       | Second principal y cosine                    | 18-95                       | Items 5 through 17 repeated for 6 corners |                             |
|                                     | 19                       | Third principal y cosine                     |                             |   |                             |
|                                     | 20 <sup>1</sup>          | Normal z                                     |                             |   |                             |
|                                     | 21 <sup>1</sup>          | Shear zx                                     |                             |   |                             |
|                                     | 22                       | Third principal                              |                             |   |                             |
|                                     | 23                       | First principal z cosine                     |                             |   |                             |
|                                     | 24                       | Second principal z cosine                    |                             |   |                             |
|                                     | 25                       | Third principal z cosine                     |                             |   |                             |
|                                     | 26-151                   | Items 5 through 25 repeated for 6 corners    |                             |   |                             |
| <b>CPENTA (91)</b><br><br>Nonlinear | 2                        | Stress coordinate system                     |                             |   |                             |
|                                     | 3                        | Grid/Gauss                                   |                             |   |                             |
|                                     | 4                        | Number of active points                      |                             |   |                             |
|                                     | 5                        | External grid ID<br>(0=center)               |                             |   |                             |
|                                     | 6                        | Normal x stress                              |                             |   |                             |
|                                     | 7                        | Normal y stress                              |                             |   |                             |
|                                     | 8                        | Normal z stress                              |                             |   |                             |
|                                     | 9                        | Shear xy stress                              |                             |   |                             |
|                                     | 10                       | Shear yz stress                              |                             |   |                             |
|                                     | 11                       | Shear zx stress                              |                             |   |                             |
|                                     | 12                       | Equivalent stress                            |                             | Not applicable                            |                             |
|                                     | 13                       | Effective plastic strain                     |                             |   |                             |
|                                     | 14                       | Effective creep strain                       |                             |   |                             |
|                                     | 15                       | Normal x strain                              |                             |   |                             |
|                                     | 16                       | Normal y strain                              |                             |   |                             |
|                                     | 17                       | Normal z strain                              |                             |   |                             |
|                                     | 18                       | Shear xy strain                              |                             |   |                             |
|                                     | 19                       | Shear yz strain                              |                             |   |                             |
|                                     | 20                       | Shear zx strain                              |                             |   |                             |
|                                     | 21-116                   | Items 5 through 20<br>Repeated for 6 corners |                             |   |                             |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |   | Complex Stresses or Strains |                |                             |
|--|--------------------------|---|-----------------------------|----------------|-----------------------------|
|  | Item<br>Code             | Item  | Item<br>Code                | Item           | Real/Mag. or<br>Imag./Phase |
| <b>CPENTA FD (204)</b><br><br>Nonlinear Finite Deformation with 6 grid points  | 2-17                     | Same as CHEXA FD (201)                          |                             |                |                             |
|  | 18-92                    | Items 3 through 17 repeated for 5 Gauss points  |                             | Not applicable |                             |
| <b>CPENTA FD (209)</b><br><br>Nonlinear Finite Deformation with 15 grid points | 2-17                     | Same as CHEXA FD (201)                          |                             |                |                             |
|  | 18-317                   | Items 3 through 17 repeated for 20 Gauss points |                             | Not applicable |                             |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)     | Real Stresses or Strains |   | Complex Stresses or Strains |   | Real/Mag. or<br>Imag./Phase |
|----------------------------|--------------------------|---|-----------------------------|---|-----------------------------|
|                            | Item<br>Code             | Item                                      | Item<br>Code                | Item                                      |                             |
| CPYRAM (242)<br><br>Linear | 2                        | Stress coordinate system                  | 2                           | Stress coordinate system                  |                             |
|                            | 3                        | Coordinate Type (Character)               | 3                           | Coordinate Type (Character)               |                             |
|                            | 4                        | Number of active points                   | 4                           | Number of active points                   |                             |
|                            | 5                        | External grid ID (0=centre)               | 5                           | External grid ID (0=centre)               |                             |
|                            | 6                        | Normal x                                  | 6                           | Normal x                                  | RM                          |
|                            | 7                        | Shear xy                                  | 7                           | Normal y                                  | RM                          |
|                            | 8                        | First principal                           | 8                           | Normal z                                  | RM                          |
|                            | 9                        | First principal x cosine                  | 9                           | Shear xy                                  | RM                          |
|                            | 10                       | Second principal x cosine                 | 10                          | Shear yz                                  | RM                          |
|                            | 11                       | Third principal x cosine                  | 11                          | Shear zx                                  | RM                          |
|                            | 12                       | Mean pressure                             | 12                          | Normal x                                  | IP                          |
|                            | 13                       | von Mises or octahedral shear stress      | 13                          | Normal y                                  | IP                          |
|                            | 14                       | Normal y                                  | 14                          | Normal z                                  | IP                          |
|                            | 15                       | Shear yz                                  | 15                          | Shear xy                                  | IP                          |
|                            | 16                       | Second principal                          | 16                          | Shear yz                                  | IP                          |
|                            | 17                       | First principal y cosine                  | 17                          | Shear zx                                  | IP                          |
|                            | 18                       | Second principal y cosine                 | 18-82                       | Items 5 through 17 repeated for 5 corners |                             |
|                            | 19                       | Third principal y cosine                  |                             |   |                             |
|                            | 20                       | Normal z                                  |                             |   |                             |
|                            | 21                       | Shear zx                                  |                             |   |                             |
|                            | 22                       | Third Principal                           |                             |   |                             |
|                            | 23                       | First principal z cosine                  |                             |   |                             |
|                            | 24                       | Second principal z cosine                 |                             |   |                             |
|                            | 25                       | Third principal z cosine                  |                             |   |                             |
|                            | 26-130                   | Items 5 through 25 repeated for 5 corners |                             |   |                             |

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

| Element Name<br>(Code)                | Real Stresses or Strains |   | Complex Stresses or Strains |      | Real/Mag. or<br>Imag./Phase |
|---------------------------------------|--------------------------|---|-----------------------------|------|-----------------------------|
|                                       | Item<br>Code             | Item                                      | Item<br>Code                | Item |                             |
| <b>CPYRAM (242)</b><br><br>Non Linear | 2                        | Stress coordinate system                  | Not Applicable              |      |                             |
|                                       | 3                        | Grid/Gauss                                |                             |      |                             |
|                                       | 4                        | Number of active points                   |                             |      |                             |
|                                       | 5                        | External grid ID (0=centre)               |                             |      |                             |
|                                       | 6                        | Stress X                                  |                             |      |                             |
|                                       | 7                        | Stress Y                                  |                             |      |                             |
|                                       | 8                        | Stress Z                                  |                             |      |                             |
|                                       | 9                        | Stress XY                                 |                             |      |                             |
|                                       | 10                       | Stress YZ                                 |                             |      |                             |
|                                       | 11                       | Stress ZX                                 |                             |      |                             |
|                                       | 12                       | Equivalent Stress                         |                             |      |                             |
|                                       | 13                       | Effective plastic strain                  |                             |      |                             |
|                                       | 14                       | Effective creep strain                    |                             |      |                             |
|                                       | 15                       | Strain X                                  |                             |      |                             |
|                                       | 16                       | Strain Y                                  |                             |      |                             |
|                                       | 17                       | Strain Z                                  |                             |      |                             |
|                                       | 18                       | Strain XY                                 |                             |      |                             |
|                                       | 19                       | Strain YZ                                 |                             |      |                             |
|                                       | 20                       | Strain ZX                                 |                             |      |                             |
|                                       | 21-100                   | Items 5 through 20 repeated for 5 corners |                             |      |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3

| Element Name<br>(Code)           | Real Stresses or Strains |                     | Complex Stresses or Strains |                     | Real/Mag. or<br>Imag./Phase |
|----------------------------------|--------------------------|---------------------|-----------------------------|---------------------|-----------------------------|
|                                  | Item<br>Code             | Item                | Item<br>Code                | Item                |                             |
| <b>CQUAD4 (33)</b><br><br>Linear | 2                        | Z1=Fiber distance 1 | 2                           | Z1=Fiber distance 1 | RM                          |
|                                  | 3 <sup>1</sup>           | Normal x at Z1      | 3 <sup>1</sup>              | Normal x at Z1      |                             |
|                                  | 4 <sup>1</sup>           | Normal y at Z1      | 4 <sup>1</sup>              | Normal x at Z1      |                             |
|                                  | 5 <sup>1</sup>           | Shear xy at Z1      | 5 <sup>1</sup>              | Normal y at Z1      |                             |
|                                  | 6                        | Shear angle at Z1   | 6 <sup>1</sup>              | Normal y at Z1      |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)                | Real Stresses or Strains |   | Complex Stresses or Strains |                     | Real/Mag. or<br>Imag./Phase |
|---------------------------------------|--------------------------|---|-----------------------------|---------------------|-----------------------------|
|                                       | Item<br>Code             | Item  | Item<br>Code                | Item                |                             |
|                                       | 7                        | Major principal at Z1   | 7 <sup>1</sup>              | Shear xy at Z1      | RM                          |
|                                       | 8                        | Minor principal at Z1   | 8 <sup>1</sup>              | Shear xy at Z1      | IP                          |
|                                       | 9                        | von Mises or<br>maximum shear at Z1   | 9                           | Z2=Fiber distance 2 |                             |
|                                       | 10                       | Z2=Fiber distance 2   | 10 <sup>1</sup>             | Normal x at Z2      | RM                          |
|                                       | 11 <sup>1</sup>          | Normal x at Z2  | 11 <sup>1</sup>             | Normal x at Z2      | IP                          |
|                                       | 12 <sup>1</sup>          | Normal y at Z2  | 12 <sup>1</sup>             | Normal y at Z2      | RM                          |
|                                       | 13 <sup>1</sup>          | Shear xy at Z2  | 13 <sup>1</sup>             | Normal y at Z2      | IP                          |
|                                       | 14                       | Shear angle at Z2   | 14                          | Shear xy at Z2      | RM                          |
|                                       | 15                       | Major principal at Z2   | 15                          | Shear xy at Z2      | IP                          |
|                                       | 16                       | Minor principal at Z2   |                             |                     |                             |
|                                       | 17                       | von Mises or<br>maximum shear at Z2   |                             |                     |                             |
| CQUAD4 (90)<br>Nonlinear              | 2                        | Z1=Fiber distance 1 (plane<br>stress only)                                  |                             |                     |                             |
|                                       | 3                        | Stress-X (at Z1, if plane stress)   |                             |                     |                             |
|                                       | 4                        | Stress-Y (at Z1, if plane stress)   |                             |                     |                             |
|                                       | 5                        | Stress-Z (plane strain only)  |                             |                     |                             |
|                                       | 6                        | Stress-XY (at Z1, if plane stress)  |                             |                     |                             |
|                                       | 7                        | Equivalent stress (at Z1, if<br>plane stress)                               |                             |                     |                             |
|                                       | 8                        | Plastic strain (at Z1, if plane<br>stress)                                  |                             |                     |                             |
|                                       | 9                        | Creep strain (at Z1, if plane<br>stress)                                    |                             | Not applicable      |                             |
|                                       | 10                       | Strain-X (at Z1, if plane stress)   |                             |                     |                             |
|                                       | 11                       | Strain-Y (at Z1, if plane stress)   |                             |                     |                             |
|                                       | 12                       | Strain-Z (plane strain only)  |                             |                     |                             |
|                                       | 13                       | Strain-XY (at Z1, if plane<br>stress)                                       |                             |                     |                             |
|                                       | 14-25                    | Items 2 through 13 repeated for<br>fiber distance Z2 (plane stress<br>only) |                             |                     |                             |
| CQUAD4 <sup>2</sup> (95)<br>Composite | 2                        | Lamina Number   | 2                           | Lamina Number       |                             |
|                                       | 3                        | Normal-1  | 3                           | Normal-1            | RM                          |
|                                       | 4                        | Normal-2  | 4                           | Normal-2            | RM                          |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)           | Real Stresses or Strains |                                   | Complex Stresses or Strains |                                   | Real/Mag. or<br>Imag./Phase |
|----------------------------------|--------------------------|-----------------------------------|-----------------------------|-----------------------------------|-----------------------------|
|                                  | Item<br>Code             | Item                              | Item<br>Code                | Item                              |                             |
|                                  | 5                        | Shear-12                          | 5                           | Shear-12                          | RM                          |
|                                  | 6                        | Shear-1Z                          | 6                           | Shear-1Z                          | RM                          |
|                                  | 7                        | Shear-2Z                          | 7                           | Shear-2Z                          | RM                          |
|                                  | 8                        | Shear angle                       | 8                           | Normal-1                          | IP                          |
|                                  | 9                        | Major principal                   | 9                           | Normal-1                          | IP                          |
|                                  | 10                       | Minor principal                   | 10                          | Shear-12                          | IP                          |
|                                  | 11                       | Maximum shear                     | 11                          | Shear-1Z                          | IP                          |
|                                  | 12                       |                                   | 12                          | Shear-2Z                          | IP                          |
| CQUAD4 (144)<br>CORNER<br>Output | 1                        | EID                               | 1                           | EID                               |                             |
|                                  | 2                        | CEN/                              | 2                           | CEN/                              |                             |
|                                  | 3                        | 4                                 | 3                           | 4                                 |                             |
|                                  | 4                        | Z1-Fiber distance                 | 4                           | Z1-Fiber distance                 |                             |
|                                  | 5                        | Normal x at Z1                    | 5                           | Normal x at Z1                    | RM                          |
|                                  | 6                        | Normal y at Z1                    | 6                           | Normal x at Z1                    | IP                          |
|                                  | 7                        | Shear xy at Z1                    | 7                           | Normal y at Z1                    | RM                          |
|                                  | 8                        | Shear angle at Z1                 | 8                           | Normal y at Z1                    | IP                          |
|                                  | 9                        | Major principal at Z1             | 9                           | Shear xy at Z1                    | RM                          |
|                                  | 10                       | Minor principal at Z1             | 10                          | Shear xy at Z1                    | IP                          |
|                                  | 11                       | von Mises or maximum shear at Z1  | 11                          | Z2-Fiber distance                 |                             |
|                                  | 12                       | Z2-Fiber distance                 | 12                          | Normal x at Z2                    | RM                          |
|                                  | 13                       | Normal x at Z2                    | 13                          | Normal x at Z2                    | IP                          |
|                                  | 14                       | Normal y at Z2                    | 14                          | Normal y at Z2                    | RM                          |
|                                  | 15                       | Shear xy at Z2                    | 15                          | Normal y at Z2                    | IP                          |
|                                  | 16                       | Shear angle at Z2                 | 16                          | Shear xy at Z2                    | RM                          |
|                                  | 17                       | Major principal at Z2             | 17                          | Shear xy at Z2                    | IP                          |
|                                  | 18                       | Minor principal at Z2             | 18                          | Grid 1                            |                             |
|                                  | 19                       | von Mises or maximum shear at Z2  | 19-32                       | Same as 4 through 17 for corner 1 |                             |
|                                  | 20                       | Grid 1                            | 33                          | Grid 2                            |                             |
|                                  | 21-36                    | Same as 4 through 19 for corner 1 | 34-47                       | Same as 4 through 17 for corner 2 |                             |
|                                  | 37                       | Grid 2                            | 48                          | Grid 3                            |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)                | Real Stresses or Strains |   | Complex Stresses or Strains |   | Real/Mag. or<br>Imag./Phase |
|---------------------------------------|--------------------------|---|-----------------------------|---|-----------------------------|
|                                       | Item<br>Code             | Item                                    | Item<br>Code                | Item                                    |                             |
|                                       | 38-53                    | Same as 4 through 19 for corner 2       | 49-62                       | Same as 4 through 17 for corner 3       |                             |
|                                       | 54                       | Grid 3                                  | 63                          | Grid 4                                  |                             |
|                                       | 55-70                    | Same as 4 through 19 for corner 3       | 64-77                       | Same as 4 through 17 for corner 4       |                             |
|                                       | 71                       | Grid 4                                  |                             |   |                             |
|                                       | 72-87                    | Same as 4 through 19 for corner 4       |                             |   |                             |
| CQUAD8 (64)                           | 5 <sup>1</sup>           | Normal x at Z1                          | 5 <sup>1</sup>              | Normal x at Z1                          | RM                          |
|                                       | 6 <sup>1</sup>           | Normal y at Z1                          | 6 <sup>1</sup>              | Normal x at Z1                          | IP                          |
|                                       | 7 <sup>1</sup>           | Shear xy at Z1                          | 7 <sup>1</sup>              | Normal y at Z1                          | RM                          |
|                                       | 8                        | O Shear angle at Z1                     | 8 <sup>1</sup>              | Normal y at Z1                          | IP                          |
|                                       | 9                        | Major principal at Z1                   | 9 <sup>1</sup>              | Shear xy at Z1                          | RM                          |
|                                       | 10                       | Minor principal at Z1                   | 10 <sup>1</sup>             | Shear xy at Z1                          | IP                          |
|                                       | 11                       | von Mises or maximum shear at Z1        | 12 <sup>1</sup>             | Normal x at Z2                          | RM                          |
|                                       | 13 <sup>1</sup>          | Normal x at Z2                          | 13 <sup>1</sup>             | Normal x at Z2                          | IP                          |
|                                       | 14 <sup>1</sup>          | Normal y at Z2                          | 14 <sup>1</sup>             | Normal y at Z2                          | RM                          |
|                                       | 15 <sup>1</sup>          | Shear xy at Z2                          | 15 <sup>1</sup>             | Normal y at Z2                          | IP                          |
|                                       | 16                       | Q Shear angle at Z2                     | 16 <sup>1</sup>             | Shear xy at Z2                          | RM                          |
|                                       | 17                       | Major principal at Z2                   | 17 <sup>1</sup>             | Shear xy at Z2                          | IP                          |
|                                       | 18                       | Minor principal at Z2                   | 20-32                       | Same as items 5 through 17 for corner 1 |                             |
|                                       | 19                       | von Mises or maximum shear at Z2        | 35-47                       | Same as items 5 through 17 for corner 2 |                             |
|                                       | 22-36                    | Same as items 5 through 19 for corner 1 | 50-62                       | Same as items 5 through 17 for corner 3 |                             |
|                                       | 39-53                    | Same as items 5 through 19 for corner 2 | 65-77                       | Same as items 5 through 17 for corner 4 |                             |
|                                       | 56-70                    | Same as items 5 through 19 for corner 3 |                             |   |                             |
|                                       | 73-87                    | Same as items 5 through 19 for corner 4 |                             |   |                             |
| CQUAD8 <sup>2</sup> (96)<br>Composite |                          | Same as CQUAD4(95)                      |                             | Same as CQUAD4(95)                      |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |  | Complex Stresses or Strains |                     | Real/Mag. or<br>Imag./Phase |
|--|--------------------------|--|-----------------------------|---------------------|-----------------------------|
|  | Item<br>Code             | Item   | Item<br>Code                | Item                |                             |
| <b>CQUADFD (201)</b><br><br>Nonlinear Finite Deformation with 4 grid points      | 2                        | Grid/Gauss                                     |                             |                     |                             |
|  | 3                        | Grid/Gauss ID (0=center)                       |                             |                     |                             |
|  | 4                        | Cauchy stress-X                                |                             |                     |                             |
|  | 5                        | Cauchy stress-Y                                |                             |                     |                             |
|  | 6                        | Cauchy stress-Z                                |                             |                     |                             |
|  | 7                        | Cauchy stress-XY                               |                             |                     |                             |
|  | 8                        | $\frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$  |                             | Not applicable      |                             |
|  | 9                        | Volumetric strain J-1                          |                             |                     |                             |
|  | 10                       | Logarithmic strain-X                           |                             |                     |                             |
|  | 11                       | Logarithmic strain-Y                           |                             |                     |                             |
|  | 13                       | Logarithmic strain-XY                          |                             |                     |                             |
|  | 14-46                    | Items 3 through 13 repeated for 3 Gauss points |                             |                     |                             |
| <b>CQUADFD (208)</b><br><br>Nonlinear Finite Deformation with 8 or 9 grid points | 2-13                     | Same as CQUADFD (201)                          |                             |                     |                             |
|  | 14-101                   | Items 3 through 13 repeated for 8 Gauss points |                             | Not applicable      |                             |
| <b>CQUADR (82)</b><br><br>Linear   |                          | Same as CQUAD4(144)                            |                             | Same as CQUAD4(144) |                             |
| <b>CQUADR (172)</b><br><br>Nonlinear   |                          | Same as CQUAD4(90)                             |                             | Not applicable      |                             |
| <b>CQUADR (232)</b><br><br>Composite   |                          | Same as CQUAD4(95)                             |                             | Same as CQUAD4(95)  |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |  | Complex Stresses or Strains |                | Real/Mag. or<br>Imag./Phase |
|---|--------------------------|--|-----------------------------|----------------|-----------------------------|
|   | Item<br>Code             | Item   | Item<br>Code                | Item           |                             |
| CQUADX(18)<br><br>Harmonic  | 1                        | (Elem ID)  | 1                           | (Elem ID)      |                             |
|   | 2                        | Harmonic   | 2                           | Harmonic       |                             |
|   | 3                        | CEN/grid   | 3                           | CEN/grid       |                             |
|   | 4                        | Stress_xx  | 4                           | Stress_xx      | RM                          |
|   | 5                        | Stress_yy  | 5                           | Stress_yy      | RM                          |
|   | 6                        | Stress_tt  | 6                           | Stress_tt      | RM                          |
|   | 7                        | Stress_xy  | 7                           | Stress_xy      | RM                          |
|   | 8                        | Stress_yt  | 8                           | Stress_yt      | RM                          |
|   | 9                        | Stress_tx  | 9                           | Stress_tx      | RM                          |
|   | 10                       | VON MISES*   | 10                          | Stress_xx      | IP                          |
|   | 11-18                    | 1st corner   | 11                          | Stress_yy      | IP                          |
|   | 19-26                    | 2nd corner   | 12                          | Stress_tt      | IP                          |
|   | 27-34                    | 3rd corner   | 13                          | Stress_xy      | IP                          |
|   | 35-42                    | 4th corner   | 14                          | Stress_yt      | IP                          |
|   |                          |  | 15                          | Stress_tx      | IP                          |
|   |                          |  | 16-28                       | 1st corner     |                             |
|   |                          |  | 29-41                       | 2nd corner     |                             |
|   |                          |  | 42-54                       | 3rd corner     |                             |
|   |                          |  | 55-67                       | 4th corner     |                             |
| CQUADXF(214)<br><br>Nonlinear Finite Deformation with 4 grid points | 2                        | Grid/Gauss   |                             |                |                             |
|   | 3                        | Gauss ID   |                             |                |                             |
|   | 4                        | Cauchy stress-X (radial)                                   |                             |                |                             |
|   | 5                        | Cauchy stress-Y (axial)                                    |                             |                |                             |
|   | 6                        | Cauchy stress-Z (circumferential)                          |                             |                |                             |
|   | 7                        | Cauchy stress-XY   |                             |                |                             |
|   | 8                        | Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$ |                             | Not applicable |                             |
|   | 9                        | Volumetric strain J-1                                      |                             |                |                             |
|   | 10                       | Logarithmic strain-X (radial)                              |                             |                |                             |
|   | 11                       | Logarithmic strain-Y (axial)                               |                             |                |                             |
|   | 12                       | Logarithmic strain-Z (circumferential)                     |                             |                |                             |
|   | 13                       | Logarithmic strain-XY                                      |                             |                |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)                               | Real Stresses or Strains |  | Complex Stresses or Strains |                    |                             |
|--|--------------------------|--|-----------------------------|--------------------|-----------------------------|
|  | Item<br>Code             | Item   | Item<br>Code                | Item               | Real/Mag. or<br>Imag./Phase |
|  | 14-46                    | Items 3 through 13 repeated for remaining 3 Gauss points |                             |                    |                             |
| <b>CQUADXFD (215)</b>                                | 2-13                     | Same as QUADXFD (214)                                    |                             | Not applicable     |                             |
| Nonlinear Finite Deformation with 8 or 9 grid points | 14-101                   | Items 3 through 13 repeated for remaining 8 Gauss points |                             |                    |                             |
| <b>CROD (1)</b>                                      |                          | Same as CONROD(10)                                       |                             | Same as CONROD(10) |                             |
| Linear   |                          |  |                             |                    |                             |
| <b>CROD (89)</b>                                     |                          | Same as CONROD(92)                                       |                             | Not applicable     |                             |
| Nonlinear  |                          |  |                             |                    |                             |
| <b>CSHEAR (4)</b>                                    | 2                        | Maximum shear  | 2                           | Maximum shear      | RM                          |
|  | 3 <sup>1</sup>           | Average shear  | 3                           | Maximum shear      | IP                          |
|  | 4                        | Safety margin  | 4 <sup>1</sup>              | Average shear      | RM                          |
|  |                          |  | 5 <sup>1</sup>              | Average shear      | IP                          |
| <b>CSLOT3 (50)</b>                                   | 2                        | Radial centroid  | 2                           | Radial centroid    | RM                          |
|  | 3                        | Axial centroid   | 3                           | Axial centroid     | RM                          |
|  | 4                        | Tangential edge 1  | 4                           | Tangential edge 1  | RM                          |
|  | 5                        | Tangential edge 2  | 5                           | Tangential edge 2  | RM                          |
|  | 6                        | Tangential edge 3  | 6                           | Tangential edge 3  | RM                          |
|  |                          |  | 7                           | Radial centroid    | IP                          |
|  |                          |  | 8                           | Axial centroid     | IP                          |
|  |                          |  | 9                           | Tangential edge 1  | IP                          |
|  |                          |  | 10                          | Tangential edge 2  | IP                          |
|  |                          |  | 11                          | Tangential edge 3  | IP                          |
|  |                          |  |                             |                    |                             |
| <b>CSLOT4 (51)</b>                                   | 2                        | Radial centroid  | 2                           | Radial centroid    | RM                          |
|  | 3                        | Axial centroid   | 3                           | Axial centroid     | RM                          |
|  | 4                        | Tangential edge 1  | 4                           | Tangential edge 1  | RM                          |
|  | 5                        | Tangential edge 2  | 5                           | Tangential edge 2  | RM                          |
|  | 6                        | Tangential edge 3  | 6                           | Tangential edge 3  | RM                          |
|  | 7                        | Tangential edge 4  | 7                           | Tangential edge 4  | RM                          |
|  |                          |  | 8                           | Radial centroid    | IP                          |
|  |                          |  | 9                           | Axial centroid     | IP                          |
|  |                          |  | 10                          | Tangential edge 1  | IP                          |
|  |                          |  | 11                          | Tangential edge 2  | IP                          |
|  |                          |  |                             |                    |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code) | Real Stresses or Strains |  | Complex Stresses or Strains |  | Real/Mag. or<br>Imag./Phase |
|------------------------|--------------------------|--|-----------------------------|--|-----------------------------|
|                        | Item<br>Code             | Item   | Item<br>Code                | Item   |                             |
|                        |                          |  | 12                          | Tangential edge 3                            | IP                          |
|                        |                          |  | 13                          | Tangential edge 4                            | IP                          |
| CTETRA (39)            | 2                        | Stress coordinate system                     | 2                           | Stress coordinate system                     |                             |
| Linear                 | 3                        | Coordinate type (Character)                  | 3                           | Coordinate type (Character)                  |                             |
|                        | 4                        | Number of active points                      | 4                           | Number of active points                      |                             |
|                        | 5                        | External grid ID                             | 5                           | External grid ID                             |                             |
|                        |                          | (0=center)                                   |                             | (0=center)                                   |                             |
|                        | 6 <sup>1</sup>           | Normal x                                     | 6 <sup>1</sup>              | Normal x                                     | RM                          |
|                        | 7 <sup>1</sup>           | Shear xy                                     | 7 <sup>1</sup>              | Normal y                                     | RM                          |
|                        | 8                        | First principal                              | 8 <sup>1</sup>              | Normal z                                     | RM                          |
|                        | 9                        | First principal x cosine                     | 9 <sup>1</sup>              | Shear xy                                     | RM                          |
|                        | 10                       | Second principal x cosine                    | 10 <sup>1</sup>             | Shear yz                                     | RM                          |
|                        | 11                       | Third principal x cosine                     | 11 <sup>1</sup>             | Shear zx                                     | RM                          |
|                        | 12                       | Mean pressure                                | 12 <sup>1</sup>             | Normal x                                     | IP                          |
|                        | 13                       | von Mises or octahedral shear stress         | 13 <sup>1</sup>             | Normal y                                     | IP                          |
|                        | 14 <sup>1</sup>          | Normal y                                     | 14 <sup>1</sup>             | Normal z                                     | IP                          |
|                        | 15 <sup>1</sup>          | Shear yz                                     | 15 <sup>1</sup>             | Shear xy                                     | IP                          |
|                        | 16                       | Second principal                             | 16 <sup>1</sup>             | Shear yz                                     | IP                          |
|                        | 17                       | First principal y cosine                     | 17 <sup>1</sup>             | Shear zx                                     | IP                          |
|                        | 18                       | Second principal y cosine                    | 18-69                       | Items 5 through 17 repeated for four corners |                             |
|                        | 19                       | Third principal y cosine                     |                             |  |                             |
|                        | 20 <sup>1</sup>          | Normal z                                     |                             |  |                             |
|                        | 21 <sup>1</sup>          | Shear zx                                     |                             |  |                             |
|                        | 22                       | Third principal                              |                             |  |                             |
|                        | 23                       | First principal z cosine                     |                             |  |                             |
|                        | 24                       | Second principal z cosine                    |                             |  |                             |
|                        | 25                       | Third principal z cosine                     |                             |  |                             |
|                        | 26-109                   | Items 5 through 25 repeated for four corners |                             |  |                             |

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |   | Complex Stresses or Strains |                |                             |
|---|--------------------------|---|-----------------------------|----------------|-----------------------------|
|   | Item<br>Code             | Item  | Item<br>Code                | Item           | Real/Mag. or<br>Imag./Phase |
| <b>CTETRA (85)</b><br>Nonlinear   | 2                        | Stress coordinate system                          |                             |                |                             |
|   | 3                        | Grid/Gauss  |                             |                |                             |
|   | 4                        | Number of active points                           |                             |                |                             |
|   | 5                        | External grid ID<br>(0=center)                    |                             |                |                             |
|   | 6                        | Stress-X  |                             |                |                             |
|   | 7                        | Stress-Y  |                             |                |                             |
|   | 8                        | Stress-Z  |                             |                |                             |
|   | 9                        | Stress-XY   |                             |                |                             |
|   | 10                       | Stress-YZ   |                             |                |                             |
|   | 11                       | Stress-ZX   |                             |                |                             |
|   | 12                       | Equivalent stress                                 |                             |                |                             |
|   | 13                       | Effective plastic strain                          |                             | Not applicable |                             |
|   | 14                       | Effective creep strain                            |                             |                |                             |
|   | 15                       | Strain-X  |                             |                |                             |
|   | 16                       | Strain-Y  |                             |                |                             |
|   | 17                       | Strain-Z  |                             |                |                             |
|   | 18                       | Strain-XY   |                             |                |                             |
|   | 19                       | Strain-YZ   |                             |                |                             |
|   | 20                       | Strain-ZX   |                             |                |                             |
|   | 21-84                    | Items 5 through 20<br>Repeated for four corners   |                             |                |                             |
| <b>CTETRAFD (205)</b><br>Nonlinear Finite<br>Deformation with<br>4 grid points  | 2-17                     | Same as CHEXA FD (202)                            |                             | Not applicable |                             |
| <b>CTETRAFD (210)</b><br>Nonlinear Finite<br>Deformation with<br>10 grid points | 2-17                     | Same as CHEXA FD (202)                            |                             |                |                             |
|   | 18-77                    | Items 3 through 17 repeated for<br>4 Gauss points |                             | Not applicable |                             |

Table 7-4 Element Stress-Strain Item Codes Part 4

| Element Name<br>(Code)  | Real Stresses or Strains |  | Complex Stresses or Strains |  | Real/Mag. or<br>Imag./Phase |
|---|--------------------------|--|-----------------------------|--|-----------------------------|
|   | Item<br>Code             | Item                                       | Item<br>Code                | Item                                       |                             |
| CTRIA3 (74)   |                          | Same as CQUAD4(33)                         |                             | Same as CQUAD4(33)                         |                             |
| CTRIA3 <sup>2</sup> (97)<br>Composite                           |                          | Same as CQUAD4(95)                         |                             | Same as CQUAD4(95)                         |                             |
| CTRIA3 (88)<br>Nonlinear  |                          | Same as CQUAD4(90)                         |                             | Not applicable                             |                             |
| CTRIA6 (75)<br>Linear   | 5 <sup>1</sup>           | Normal x at Z1                             | 5 <sup>1</sup>              | Normal x at Z1                             | RM                          |
|   | 6 <sup>1</sup>           | Normal y at Z1                             | 6 <sup>1</sup>              | Normal x at Z1                             | IP                          |
|   | 7 <sup>1</sup>           | Shear xy at Z1                             | 7 <sup>1</sup>              | Normal y at Z1                             | RM                          |
|   | 8                        | Q shear angle at Z1                        | 8 <sup>1</sup>              | Normal y at Z1                             | IP                          |
|   | 9                        | Major principal at Z1                      | 9 <sup>1</sup>              | Shear xy at Z1                             | RM                          |
|   | 10                       | Minor principal at Z1                      | 10 <sup>1</sup>             | Shear xy at Z1                             | IP                          |
|   | 11                       | von Mises or<br>maximum shear at Z1        | 12 <sup>1</sup>             | Normal x at Z2                             | RM                          |
|   | 13 <sup>1</sup>          | Normal x at Z2                             | 13 <sup>1</sup>             | Normal x at Z2                             | IP                          |
|   | 14 <sup>1</sup>          | Normal y at Z2                             | 14 <sup>1</sup>             | Normal y at Z2                             | RM                          |
|   | 15 <sup>1</sup>          | Shear xy at Z2                             | 15 <sup>1</sup>             | Normal y at Z2                             | IP                          |
|   | 16                       | Q shear angle at Z2                        | 16 <sup>1</sup>             | Shear xy at Z2                             | RM                          |
|   | 17                       | Major principal at Z2                      | 17 <sup>1</sup>             | Shear xy at Z2                             | IP                          |
|   | 18                       | Minor principal at Z2                      | 20-32                       | Same as items 5 through 17<br>for corner 1 |                             |
|   | 19                       | von Mises or<br>maximum shear at Z2        | 35-47                       | Same as items 5 through 17<br>for corner 2 |                             |
|   | 22-36                    | Same as items 5 through 19<br>for corner 1 | 50-62                       | Same as items 5 through 17<br>for corner 3 |                             |
|   | 39-53                    | Same as items 5 through 19<br>for corner 2 |                             |  |                             |
|   | 56-70                    | Same as items 5 through 19<br>for corner 3 |                             |  |                             |
| CTRIA6 <sup>2</sup> (98)  |                          | Same as CQUAD4(95)                         |                             | Same as CQUAD4(95)                         |                             |
| CTRIAFD (206)<br>Nonlinear<br>Deformation with 3<br>grid points | 2-13                     | Same as CQUADFD(201)                       |                             | Not applicable                             |                             |

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |  | Complex Stresses or Strains |                    | Real/Mag. or<br>Imag./Phase |
|--|--------------------------|--|-----------------------------|--------------------|-----------------------------|
|  | Item<br>Code             | Item   | Item<br>Code                | Item               |                             |
| CTRIAFD (211)<br>Nonlinear Finite Deformation with 6 grid points | 2-13                     | Same as CQUADFD (201)                          |                             |                    |                             |
|  | 14-35                    | Items 3 through 12 repeated for 2 Gauss points |                             | Not applicable     |                             |
| CTRIAR (70)<br>Linear  |                          | Same as CTRIA6(75)                             |                             | Same as CTRIA6(75) |                             |
| CTRIAR (173)<br>Nonlinear  |                          | Same as CQUAD4(90)                             |                             | Not applicable     |                             |
| CTRIAR (233)<br>Composite  |                          | Same as CQUAD4(95)                             |                             | Same as CQUAD4(95) |                             |
| CTRIAX(17)<br>Harmonic   | 1                        | (Elem ID)                                      | 1                           | (Elem ID)          |                             |
|  | 2                        | Harmonic                                       | 2                           | Harmonic           |                             |
|  | 3                        | CEN/grid                                       | 3                           | CEN/grid           |                             |
|  | 4                        | Stress_xx                                      | 4                           | Stress_xx          | RM                          |
|  | 5                        | Stress_yy                                      | 5                           | Stress_yy          | RM                          |
|  | 6                        | Stress_tt                                      | 6                           | Stress_tt          | RM                          |
|  | 7                        | Stress_xy                                      | 7                           | Stress_xy          | RM                          |
|  | 8                        | Stress_yt                                      | 8                           | Stress_yt          | RM                          |
|  | 9                        | Stress_tx                                      | 9                           | Stress_tx          | RM                          |
|  | 10                       | VON MISES*                                     | 10                          | Stress_xx          | IP                          |
|  | 11-18                    | 1st corner                                     | 11                          | Stress_yy          | IP                          |
|  | 19-26                    | 2nd corner                                     | 12                          | Stress_tt          | IP                          |
|  | 27-34                    | 3rd corner                                     | 13                          | Stress_xy          | IP                          |
|  |                          |  | 14                          | Stress_yt          | IP                          |
|  |                          |  | 15                          | Stress_tx          | IP                          |
|  |                          |  | 16-28                       | 1st corner         |                             |
|  |                          |  | 29-41                       | 2nd corner         |                             |
|  |                          |  | 42-54                       | 3rd corner         |                             |
| CTRIAX6 (53)   | 3 <sup>1</sup>           | Radial   | 3 <sup>1</sup>              | Radial             | RM                          |
|  | 4 <sup>1</sup>           | Azimuthal                                      | 4 <sup>1</sup>              | Radial             | IP                          |
|  | 5 <sup>1</sup>           | Axial  | 5 <sup>1</sup>              | Azimuthal          | RM                          |
|  | 6 <sup>1</sup>           | Shear stress                                   | 6 <sup>1</sup>              | Azimuthal          | IP                          |

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |   | Complex Stresses or Strains |  |                             |
|--|--------------------------|---|-----------------------------|--|-----------------------------|
|  | Item<br>Code             | Item  | Item<br>Code                | Item                                       | Real/Mag. or<br>Imag./Phase |
|  | 7                        | Maximum principal                                 | 7                           | Axial                                      | RM                          |
|  | 8                        | Maximum shear                                     | 8                           | Axial                                      | IP                          |
|  | 9                        | von Mises or octahedral                           | 9                           | Shear                                      | RM                          |
|  |                          |   | 10                          | Shear                                      | IP                          |
|  | 11-17                    | Same as items 3 through 9<br>for corner 1         | 12-19                       | Same as items 3 through 10<br>for corner 1 |                             |
|  | 19-25                    | Same as Items 3 through 9<br>for corner 2         | 21-28                       | Same as items 3 through 10<br>for corner 2 |                             |
|  | 27-33                    | Same as items 3 through 9<br>for corner 3         | 30-37                       | Same as items 3 through 10<br>for corner 3 |                             |
| <b>CTRIAXFD (212)</b><br><br>Nonlinear Finite Deformation with 3 grid points | 2-13                     | Same as CQUADXF D (214)                           |                             | Not applicable                             |                             |
| <b>CTRIAXFD (213)</b><br><br>Nonlinear Finite Deformation with 6 grid points | 2-13                     | Same as CQUADXF D (214)                           |                             |  |                             |
| <b>CTUBE (3)</b><br><br>Linear   | 14-35                    | Items 3 through 13 repeated<br>for 2 Gauss points |                             | Not applicable                             |                             |
| <b>CTUBE (87)</b><br><br>Nonlinear   |                          | Same as CONROD(92)                                |                             | Not applicable                             |                             |

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |  | Complex Stresses or Strains |  |                             |
|--|--------------------------|--|-----------------------------|--|-----------------------------|
|  | Item<br>Code             | Item   | Item<br>Code                | Item   | Real/Mag. or<br>Imag./Phase |
| <b>CWELDP</b><br><b>(118)</b><br>if PARTPAT<br>or ELPAT            | 2                        | Axial  | 2                           | Axial  | RM                          |
|  | 3                        | Maximum End A                                | 3                           | Maximum End A                                | RM                          |
|  | 4                        | Minimum End A                                | 4                           | Minimum End A                                | RM                          |
|  | 5                        | Maximum End B                                | 5                           | Maximum End B                                | RM                          |
|  | 6                        | Minimum End B                                | 6                           | Minimum End B                                | RM                          |
|  | 7                        | Maximum Shear                                | 7                           | Maximum Shear                                | RM                          |
|  | 8                        | Bearing Stress<br>(Not applicable to Strain) | 8                           | Bearing Stress<br>(Not applicable to Strain) | RM                          |
|  |                          |  | 9                           | Axial  | IP                          |
|  |                          |  | 10                          | Maximum End A                                | IP                          |
|  |                          |  | 11                          | Minimum End A                                | IP                          |
|  |                          |  | 12                          | Maximum End B                                | IP                          |
|  |                          |  | 13                          | Minimum End B                                | IP                          |
|  |                          |  | 14                          | Maximum Shear                                | IP                          |
|  |                          |  | 15                          | Bearing Stress<br>(Not applicable to Strain) | IP                          |
| <b>CWELDC</b><br><b>(117)</b><br>if<br>MSET=OFF and<br>oldweld=yes | 1                        | VU element ID * 10 + device<br>code          |                             |  |                             |
|  | 2                        | Parent p-element ID                          |                             |  |                             |
|  | 3                        | VU grid ID for corner 1                      |                             |  |                             |
|  | 4                        | Normal x                                     | 4                           | Normal x                                     | RM                          |
|  | 5                        | Normal y                                     | 5                           | Normal y                                     | RM                          |
|  | 6                        | Normal z                                     | 6                           | Normal z                                     | RM                          |
|  | 7                        | Shear xy                                     | 7                           | Shear xy                                     | RM                          |
|  | 8                        | Shear yz                                     | 8                           | Shear yz                                     | RM                          |
|  | 9                        | Shear zx                                     | 9                           | Shear zx                                     | RM                          |
|  | 10                       | First principal                              | 10                          | Normal x                                     | IP                          |
|  | 11                       | Second principal                             | 11                          | Normal y                                     | IP                          |
|  | 12                       | Third principal                              | 12                          | Normal z                                     | IP                          |
|  | 13                       | Mean pressure                                | 13                          | Shear xy                                     | IP                          |
|  | 14                       | von Mises/ Octahedral                        | 14                          | Shear yz                                     | IP                          |
| <b>CWELD</b><br><b>(200)</b><br>if<br>MSET=ON or<br>olddweld=no    | 15-26                    | Repeat items 3-14 for corner<br>2            | 15                          | Shear zx                                     | IP                          |
|  | 1                        | VU element ID * 10 + device<br>code          |                             |  |                             |
|  | 2                        | Parent p-element ID                          |                             |  |                             |
|  | 3                        | VU grid ID for corner 1                      |                             |  |                             |
|  | 4                        | Normal x                                     | 4                           | Normal x                                     | RM                          |
|  | 5                        | Normal y                                     | 5                           | Normal y                                     | RM                          |
|  | 6                        | Normal z                                     | 6                           | Normal z                                     | RM                          |
|  | 7                        | Shear xy                                     | 7                           | Shear xy                                     | RM                          |
|  | 8                        | Shear yz                                     | 8                           | Shear yz                                     | RM                          |
|  | 9                        | Shear zx                                     | 9                           | Shear zx                                     | RM                          |
|  | 10                       | First principal                              | 10                          | Normal x                                     | IP                          |
|  | 11                       | Second principal                             | 11                          | Normal y                                     | IP                          |
|  | 12                       | Third principal                              | 12                          | Normal z                                     | IP                          |
|  | 13                       | Mean pressure                                | 13                          | Shear xy                                     | IP                          |
|  | 14                       | von Mises/ Octahedral                        | 14                          | Shear yz                                     | IP                          |
|  | 15-26                    | Repeat items 3-14 for corner<br>2            | 15                          | Shear zx                                     | IP                          |

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |  | Complex Stresses or Strains |  | Real/Mag. or<br>Imag./Phase |
|--|--------------------------|--|-----------------------------|--|-----------------------------|
|  | Item<br>Code             | Item   | Item<br>Code                | Item   |                             |
|  | 27-38                    | Repeat items 3-14 for corner 3                   | 16-28                       | Repeat items 3-15 for corner 2                   |                             |
|  | 39-50                    | Repeat items 3-14 for corner 4                   | 29-41                       | Repeat items 3-15 for corner 3                   |                             |
|  | 51-62                    | Repeat items 3-14 for corner 5 (VUPENTA, VUHEXA) | 42-54                       | Repeat items 3-15 for corner 4                   |                             |
|  | 63-74                    | Repeat items 3-14 for corner 6 (VUPENTA, VUHEXA) | 55-67                       | Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA) |                             |
|  | 75-86                    | Repeat items 3-14 for corner 7 (VUHEXA)          | 68-80                       | Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA) |                             |
|  | 87-98                    | Repeat items 3-14 for corner 8 (VUHEXA)          | 81-93                       | Repeat items 3-15 for corner 7 (VUHEXA)          |                             |
|  |                          |  | 94-106                      | Repeat items 3-15 for corner 8 (VUHEXA)          |                             |
| VUHEXA (145)<br>VUPENTA (146)<br>VUTETRA (147)<br>for HEXAp,<br>PENTAp, TETRAPt<br>if SDRPOPT=OFP<br>(no principals) | 1                        | VU element ID * 10 + device code                 |                             |  |                             |
|  | 2                        | Parent p-element ID                              |                             |  |                             |
|  | 3                        | VU grid ID for corner 1                          |                             |  |                             |
|  | 4                        | Normal x   | 4                           | Normal x   | RM                          |
|  | 5                        | Normal y   | 5                           | Normal y   | RM                          |
|  | 6                        | Normal z   | 6                           | Normal z   | RM                          |
|  | 7                        | Shear xy   | 7                           | Shear xy   | RM                          |
|  | 8                        | Shear yz   | 8                           | Shear yz   | RM                          |
|  | 9                        | Shear zx   | 9                           | Shear zx   | RM                          |
|  | 10-16                    | Repeat items 3-9 for corner 2                    | 10                          | Normal x   | IP                          |
|  | 17-23                    | Repeat items 3-9 for corner 3                    | 11                          | Normal y   | IP                          |
|  | 24-30                    | Repeat items 3-9 for corner 4                    | 12                          | Normal z   | IP                          |
|  | 31-37                    | Repeat items 3-9 for corner 5 (VUPENTA, VUHEXA)  | 13                          | Shear xy   | IP                          |
|  | 38-44                    | Repeat items 3-9 for corner 6 (VUPENTA, VUHEXA)  | 14                          | Shear yz   | IP                          |
|  | 45-51                    | Repeat items 3-9 for corner 7 (VUHEXA)           | 15                          | Shear zx   | IP                          |
|  | 52-58                    | Repeat items 3-9 for corner 8 (VUHEXA)           | 16-28                       | Repeat items 3-15 for corner 2                   |                             |
|  |                          |  | 29-41                       | Repeat items 3-15 for corner 3                   |                             |

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

| Element Name<br>(Code) | Real Stresses or Strains |      | Complex Stresses or Strains |  |                             |
|------------------------|--------------------------|------|-----------------------------|--|-----------------------------|
|                        | Item<br>Code             | Item | Item<br>Code                | Item   | Real/Mag. or<br>Imag./Phase |
|                        |                          |      | 42-54                       | Repeat items 3-15 for corner 4                   |                             |
|                        |                          |      | 55-67                       | Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA) |                             |
|                        |                          |      | 68-80                       | Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA) |                             |
|                        |                          |      | 81-93                       | Repeat items 3-15 for corner 7 (VUHEXA)          |                             |
|                        |                          |      | 94-106                      | Repeat items 3-15 for corner 8 (VUHEXA)          |                             |

Table 7-5 Element Stress-Strain Item Codes Part 5

| Element Name<br>(Code)   | Real Stresses or Strains |                                  | Complex Stresses or Strains |                |                             |
|--|--------------------------|----------------------------------|-----------------------------|----------------|-----------------------------|
|  | Item<br>Code             | Item                             | Item<br>Code                | Item           | Real/Mag. or<br>Imag./Phase |
| VUQUAD (189)<br>VUTRIA (190)<br><br>for QUADp<br>and TRIAp if<br><br>STRAIN= FIBER;<br>4th char. of<br>ICORD=<br>X Y, Z (local<br>coordinate system);<br>and SDRPORT<br>=SDRP (with<br>principals) | 1                        | VU element ID * 10 + device code |                             |                |                             |
|  | 2                        | Parent p-element ID              |                             |                |                             |
|  | 3                        | CID coordinate system ID         |                             |                |                             |
|  | 4                        | ICORD flat/curved etc.           |                             |                |                             |
|  | 5                        | THETA material angle             |                             |                |                             |
|  | 6                        | ITYPE strcur=0, fiber=1          |                             |                |                             |
|  | 7                        | VU grid ID for corner 1          |                             |                |                             |
|  | 8                        | Z1 fiber distance                |                             |                |                             |
|  | 9                        | Z2 fiber distance                |                             |                |                             |
|  | 10                       | Normal x at Z1                   | 10                          | Normal x at Z1 | RM                          |
|  | 11                       | Normal y at Z1                   | 11                          | Normal y at Z1 | RM                          |
|  | 12                       | Shear xy at Z1                   | 12                          | Shear xy at Z1 | RM                          |
|  | 13                       | Shear Angle at Z1                | 13                          | 0.0            | RM                          |
|  | 14                       | Major principal at Z1            | 14                          | 0.0            | RM                          |
|  | 15                       | Minor principal at Z1            | 15                          | 0.0            | RM                          |
|  | 16                       | vonMises/Max.Shear at Z1         | 16                          | Normal x at Z2 | RM                          |
|  | 17                       | Normal x at Z2                   | 17                          | Normal y at Z2 | RM                          |
|  | 18                       | Normal y at Z2                   | 18                          | Shear xy at Z2 | RM                          |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |   | Complex Stresses or Strains |   |                                |
|---|--------------------------|---|-----------------------------|---|--------------------------------|
|   | Item<br>Code             | Item                                    | Item<br>Code                | Item                                    | Real/Mag.<br>or<br>Imag./Phase |
|   | 19                       | Shear xy at Z2                          | 19                          | 0.0                                     | RM                             |
|   | 20                       | Shear Angle at Z2                       | 20                          | 0.0                                     | RM                             |
|   | 21                       | Major principal at Z2                   | 21                          | 0.0                                     | RM                             |
|   | 22                       | Minor principal at Z2                   | 22                          | Normal x at Z1                          | IP                             |
|   | 23                       | vonMises/Max.Shear at Z2                | 23                          | Normal y at Z1                          | IP                             |
|   |                          |   | 24                          | Shear xy at Z1                          | IP                             |
|   |                          |   | 25                          | 0.0                                     | IP                             |
|   |                          |   | 26                          | 0.0                                     | IP                             |
|   |                          |   | 27                          | 0.0                                     | IP                             |
|   |                          |   | 28                          | Normal x at Z2                          | IP                             |
|   |                          |   | 29                          | Normal y at Z2                          | IP                             |
|   |                          |   | 30                          | Shear xy at Z2                          | IP                             |
|   |                          |   | 31                          | 0.0                                     | IP                             |
|   |                          |   | 32                          | 0.0                                     | IP                             |
|   |                          |   | 33                          | 0.0                                     | IP                             |
|   | 24-40                    | Repeat items 7-23 for corner 2          | 34-60                       | Repeat items 7-33 for corner 2          |                                |
|   | 41-57                    | Repeat items 7-23 for corner 3          | 61-87                       | Repeat items 7-33 for corner 3          |                                |
|   | 58-74                    | Repeat items 7-23 for corner 4 (VUQUAD) | 88-104                      | Repeat items 7-33 for corner 4 (VUQUAD) |                                |
| <b>VUQUAD (189)<br/>VUTRIA (190)</b><br><br>for QUADp and<br>TRIAp if STRAIN=<br>FIBER; if 4th char.<br>of ICORD=X Y, Z<br>(local coordinate<br>system); and<br>SDRPOPT =OPP<br>(no principals) | 1                        | VU element ID * 10 + device code        |                             |   |                                |
|   | 2                        | Parent p-element ID                     |                             |   |                                |
|   | 3                        | CID coordinate system ID                |                             |   |                                |
|   | 4                        | ICORD flat/curved etc.                  |                             |   |                                |
|   | 5                        | THETA material angle                    |                             |   |                                |
|   | 6                        | ITYPE strcur=0, fiber=1                 |                             |   |                                |
|   | 7                        | VU grid ID for corner 1                 |                             |   |                                |
|   | 8                        | Z1 fiber distance                       |                             |   |                                |
|   | 9                        | Z2 fiber distance                       |                             |   |                                |
|   | 10                       | Normal x at Z1                          | 10                          | Normal x at Z1                          | RM                             |
|   | 11                       | Normal y at Z1                          | 11                          | Normal y at Z1                          | RM                             |
|   | 12                       | Shear xy at Z1                          | 12                          | Shear xy at Z1                          | RM                             |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code) | Real Stresses or Strains                |                | Complex Stresses or Strains |   | Real/Mag. or<br>Imag./Phase |
|------------------------|---|----------------|-----------------------------|---|-----------------------------|
|                        | Item<br>Code                            | Item           | Item<br>Code                | Item                                    |                             |
|                        | 13                                      | 0.0            | 13                          | 0.0                                     | RM                          |
|                        | 14                                      | 0.0            | 14                          | 0.0                                     | RM                          |
|                        | 15                                      | 0.0            | 15                          | 0.0                                     | RM                          |
|                        | 16                                      | 0.0            | 16                          | Normal x at Z2                          | RM                          |
|                        | 17                                      | Normal x at Z2 | 17                          | Normal y at Z2                          | RM                          |
|                        | 18                                      | Normal y at Z2 | 18                          | Shear xy at Z2                          | RM                          |
|                        | 19                                      | Shear xy at Z2 | 19                          | 0.0                                     | RM                          |
|                        | 20                                      | 0.0            | 20                          | 0.0                                     | RM                          |
|                        | 21                                      | 0.0            | 21                          | 0.0                                     | RM                          |
|                        | 22                                      | 0.0            | 22                          | Normal x at Z1                          | IP                          |
|                        | 23                                      | 0.0            | 23                          | Normal y at Z1                          | IP                          |
|                        |   |                | 24                          | Shear xy at Z1                          | IP                          |
|                        |   |                | 25                          | 0.0                                     | IP                          |
|                        |   |                | 26                          | 0.0                                     | IP                          |
|                        |   |                | 27                          | 0.0                                     | IP                          |
|                        |   |                | 28                          | Normal x at Z2                          | IP                          |
|                        |   |                | 29                          | Normal y at Z2                          | IP                          |
|                        |   |                | 30                          | Shear xy at Z2                          | IP                          |
|                        |   |                | 31                          | 0.0                                     | IP                          |
|                        |   |                | 32                          | 0.0                                     | IP                          |
|                        |   |                | 33                          | 0.0                                     | IP                          |
| 24-40                  | Repeat items 7-23 for corner 2          |                | 34-60                       | Repeat items 7-33 for corner 2          |                             |
| 41-57                  | Repeat items 7-23 for corner 3          |                | 61-87                       | Repeat items 7-33 for corner 3          |                             |
| 58-74                  | Repeat items 7-23 for corner 4 (VUQUAD) |                | 88-104                      | Repeat items 7-33 for corner 4 (VUQUAD) |                             |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |                                  | Complex Stresses or Strains |                |                                |
|---|--------------------------|----------------------------------|-----------------------------|----------------|--------------------------------|
|   | Item<br>Code             | Item                             | Item<br>Code                | Item           | Real/Mag.<br>or<br>Imag./Phase |
| VUQUAD (189)<br>VUTRIA (190)<br><br>for QUADp and<br>TRIAp if<br><br>STRAIN= FIBER<br>and 4th char. of<br>ICORD=F<br>(fixed coordinate<br>system) | 1                        | VU element ID * 10 + device code |                             |                |                                |
|   | 2                        | Parent p-element ID              |                             |                |                                |
|   | 3                        | CID coordinate system ID         |                             |                |                                |
|   | 4                        | ICORD flat/curved etc.           |                             |                |                                |
|   | 5                        | THETA material angle             |                             |                |                                |
|   | 6                        | ITYPE strcur=0, fiber=1          |                             |                |                                |
|   | 7                        | VU grid ID for corner 1          |                             |                |                                |
|   | 8                        | Z1 fiber distance                |                             |                |                                |
|   | 9                        | Z2 fiber distance                |                             |                |                                |
|   | 10                       | Normal x at Z1                   | 10                          | Normal x at Z1 | RM                             |
|   | 11                       | Normal y at Z1                   | 11                          | Normal y at Z1 | RM                             |
|   | 12                       | Shear xy at Z1                   | 12                          | Shear xy at Z1 | RM                             |
|   | 13                       | Shear yz at Z1                   | 13                          | Shear yz at Z1 | RM                             |
|   | 14                       | Shear zx at Z1                   | 14                          | Shear zx at Z1 | RM                             |
|   | 15                       | Normal z at Z1                   | 15                          | Normal z at Z1 | RM                             |
|   | 16                       | Normal x at Z2                   | 16                          | Normal x at Z2 | RM                             |
|   | 17                       | Normal y at Z2                   | 17                          | Normal y at Z2 | RM                             |
|   | 18                       | Shear xy at Z2                   | 18                          | Shear xy at Z2 | RM                             |
|   | 19                       | Shear yz at Z2                   | 19                          | Shear yz at Z2 | RM                             |
|   | 20                       | Shear zx at Z2                   | 20                          | Shear zx at Z2 | RM                             |
|   | 21                       | Normal z at Z2                   | 21                          | Normal z at Z2 | RM                             |
|   | 22                       | 0.0                              | 22                          | Normal x at Z1 | IP                             |
|   | 23                       | 0.0                              | 23                          | Normal y at Z1 | IP                             |
|   |                          |                                  | 24                          | Shear xy at Z1 | IP                             |
|   |                          |                                  | 25                          | Shear yz at Z1 | IP                             |
|   |                          |                                  | 26                          | Shear zx at Z1 | IP                             |
|   |                          |                                  | 27                          | Normal z at Z1 | IP                             |
|   |                          |                                  | 28                          | Normal x at Z2 | IP                             |
|   |                          |                                  | 29                          | Normal y at Z2 | IP                             |
|   |                          |                                  | 30                          | Shear xy at Z2 | IP                             |
|   |                          |                                  | 31                          | Shear yz at Z2 | IP                             |
|   |                          |                                  | 32                          | Shear zx at Z2 | IP                             |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code)   | Real Stresses or Strains |   | Complex Stresses or Strains |   | Real/Mag. or<br>Imag./Phase |
|--|--------------------------|---|-----------------------------|---|-----------------------------|
|  | Item<br>Code             | Item                                    | Item<br>Code                | Item                                    |                             |
|  |                          |   | 33                          | Normal z at Z2                          | IP                          |
|  | 24-40                    | Repeat items 7-23 for corner 2          | 34-60                       | Repeat items 7-33 for corner 2          |                             |
|  | 41-57                    | Repeat items 7-23 for corner 3          | 61-87                       | Repeat items 7-33 for corner 3          |                             |
|  | 58-74                    | Repeat items 7-23 for corner 4 (VUQUAD) | 88-104                      | Repeat items 7-33 for corner 4 (VUQUAD) |                             |
| <b>VUQUAD (189)<br/>VUTRIA (190)</b><br><br>for QUADp and<br>TRIAp if<br>STRAIN=STRCUR<br>and 4th char. of<br>ICORD=X Y, Z<br>(local coordinate<br>system) | 1                        | VU element ID * 10 + device code        |                             |   |                             |
|  | 2                        | Parent p-element ID                     |                             |   |                             |
|  | 3                        | CID coordinate system ID                |                             |   |                             |
|  | 4                        | ICORD flat/curved etc.                  |                             |   |                             |
|  | 5                        | THETA material angle                    |                             |   |                             |
|  | 6                        | ITYPE strcur=0, fiber=1                 |                             |   |                             |
|  | 7                        | VU grid ID for corner 1                 |                             |   |                             |
|  | 8                        | 0.0                                     |                             |   |                             |
|  | 9                        | 0.0                                     |                             |   |                             |
|  | 10                       | Membrane Strain x                       | 10                          | Membrane Strain x                       | RM                          |
|  | 11                       | Membrane Strain y                       | 11                          | Membrane Strain y                       | RM                          |
|  | 12                       | Membrane Strain xy                      | 12                          | Membrane Strain xy                      | RM                          |
|  | 13                       | 0.0                                     | 13                          | 0.0                                     | RM                          |
|  | 14                       | 0.0                                     | 14                          | 0.0                                     | RM                          |
|  | 15                       | 0.0                                     | 15                          | 0.0                                     | RM                          |
|  | 16                       | Bending Curvature x                     | 16                          | Bending Curvature x                     | RM                          |
|  | 17                       | Bending Curvature y                     | 17                          | Bending Curvature y                     | RM                          |
|  | 18                       | Bending Curvature xy                    | 18                          | Bending Curvature xy                    | RM                          |
|  | 19                       | Shear yz                                | 19                          | Shear yz                                | RM                          |
|  | 20                       | Shear zx                                | 20                          | Shear zx                                | RM                          |
|  | 21                       | 0.0                                     | 21                          | 0.0                                     | RM                          |
|  | 22                       | 0.0                                     | 22                          | Membrane Strain x                       | IP                          |
|  | 23                       | 0.0                                     | 23                          | Membrane Strain y                       | IP                          |
|  |                          |   | 24                          | Membrane Strain xy                      | IP                          |
|  |                          |   | 25                          | 0.0                                     | IP                          |
|  |                          |   | 26                          | 0.0                                     | IP                          |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code)  | Real Stresses or Strains |   | Complex Stresses or Strains |   |                                |
|---|--------------------------|---|-----------------------------|---|--------------------------------|
|   | Item<br>Code             | Item                                    | Item<br>Code                | Item                                    | Real/Mag.<br>or<br>Imag./Phase |
|   |                          |   | 27                          | 0.0                                     | IP                             |
|   |                          |   | 28                          | Bending Curvature x                     | IP                             |
|   |                          |   | 29                          | Bending Curvature y                     | IP                             |
|   |                          |   | 30                          | Bending Curvature xy                    | IP                             |
|   |                          |   | 31                          | Shear yz                                | IP                             |
|   |                          |   | 32                          | Shear zx                                | IP                             |
|   |                          |   | 33                          | 0.0                                     | IP                             |
|   | 24-40                    | Repeat items 7-23 for corner 2          | 34-60                       | Repeat items 7-33 for corner 2          |                                |
|   | 41-57                    | Repeat items 7-23 for corner 3          | 61-87                       | Repeat items 7-33 for corner 3          |                                |
|   | 58-74                    | Repeat items 7-23 for corner 4 (VUQUAD) | 88-104                      | Repeat items 7-33 for corner 4 (VUQUAD) |                                |
| <b>VUQUAD (189)<br/>VUTRIA (190)</b><br><br>for QUADp and<br>TRIAp if STRAIN=<br>STRCUR and 4th<br>char. of ICORD=F<br>(fixed coordinate<br>system) | 1                        | VU element ID *10 + device code         |                             |   |                                |
|   | 2                        | Parent p-element ID                     |                             |   |                                |
|   | 3                        | CID coordinate system ID                |                             |   |                                |
|   | 4                        | ICORD flat/curved etc.                  |                             |   |                                |
|   | 5                        | THETA material angle                    |                             |   |                                |
|   | 6                        | ITYPE strcur=0, fiber=1                 |                             |   |                                |
|   | 7                        | VU grid ID for corner 1                 |                             |   |                                |
|   | 8                        | 0.0                                     |                             |   |                                |
|   | 9                        | 0.0                                     |                             |   |                                |
|   | 10                       | Membrane Strain x                       | 10                          | Membrane Strain x                       | RM                             |
|   | 11                       | Membrane Strain y                       | 11                          | Membrane Strain y                       | RM                             |
|   | 12                       | Membrane Strain xy                      | 12                          | Membrane Strain xy                      | RM                             |
|   | 13                       | Membrane Strain yz                      | 13                          | Membrane Strain yz                      | RM                             |
|   | 14                       | Membrane Strain zx                      | 14                          | Membrane Strain zx                      | RM                             |
|   | 15                       | Membrane Strain z                       | 15                          | Membrane Strain z                       | RM                             |
|   | 16                       | Bending Curvature x                     | 16                          | Bending Curvature x                     | RM                             |
|   | 17                       | Bending Curvature y                     | 17                          | Bending Curvature y                     | RM                             |
|   | 18                       | Bending Curvature xy                    | 18                          | Bending Curvature xy                    | RM                             |
|   | 19                       | Bending Curvature yz                    | 19                          | Bending Curvature yz                    | RM                             |
|   | 20                       | Bending Curvature zx                    | 20                          | Bending Curvature zx                    | RM                             |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code)       | Real Stresses or Strains |   | Complex Stresses or Strains |   | Real/Mag. or<br>Imag./Phase |
|------------------------------|--------------------------|---|-----------------------------|---|-----------------------------|
|                              | Item<br>Code             | Item                                    | Item<br>Code                | Item                                    |                             |
|                              | 21                       | Bending Curvature z                     | 21                          | Bending Curvature z                     | RM                          |
|                              | 22                       | 0.0                                     | 22                          | Membrane Strain x                       | IP                          |
|                              | 23                       | 0.0                                     | 23                          | Membrane Strain y                       | IP                          |
|                              |                          |   | 24                          | Membrane Strain xy                      | IP                          |
|                              |                          |   | 25                          | Membrane Strain yz                      | IP                          |
|                              |                          |   | 26                          | Membrane Strain zx                      | IP                          |
|                              |                          |   | 27                          | Membrane Strain z                       | IP                          |
|                              |                          |   | 28                          | Bending Curvature x                     | IP                          |
|                              |                          |   | 29                          | Bending Curvature y                     | IP                          |
|                              |                          |   | 30                          | Bending Curvature xy                    | IP                          |
|                              |                          |   | 31                          | Bending Curvature yz                    | IP                          |
|                              |                          |   | 32                          | Bending Curvature zx                    | IP                          |
|                              |                          |   | 33                          | Bending Curvature z                     | IP                          |
|                              | 24-40                    | Repeat items 7-23 for corner 2          | 34-60                       | Repeat items 7-33 for corner 2          |                             |
|                              | 41-57                    | Repeat items 7-23 for corner 3          | 61-87                       | Repeat items 7-33 for corner 3          |                             |
|                              | 58-74                    | Repeat items 7-23 for corner 4 (VUQUAD) | 88-104                      | Repeat items 7-33 for corner 4 (VUQUAD) |                             |
| VUBEAM (191)<br>for<br>BEAMp | 1                        | VU element ID * 10 + device code        |                             |   |                             |
|                              | 2                        | Parent p-element ID                     |                             |   |                             |
|                              | 3                        | CID coordinate system ID                |                             |   |                             |
|                              | 4                        | ICORD (not used)                        |                             |   |                             |
|                              | 5                        | VU grid ID for end 1                    |                             |   |                             |
|                              | 6                        | x/L position                            |                             |   |                             |
|                              | 7                        | Y-coordinate of output point C          |                             |   |                             |
|                              | 8                        | Z-coordinate of output point C          |                             |   |                             |
|                              | 9                        | W-coordinate of output point C          |                             |   |                             |
|                              | 10                       | Normal x at C                           | 10                          | Normal x at C                           | RM                          |
|                              | 11                       | Shear xy at C                           | 11                          | Shear xy at C                           | RM                          |
|                              | 12                       | Shear zx at C                           | 12                          | Shear zx at C                           | RM                          |

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

| Element Name<br>(Code) | Real Stresses or Strains |                                      | Complex Stresses or Strains |                                      |                             |
|------------------------|--------------------------|--------------------------------------|-----------------------------|--------------------------------------|-----------------------------|
|                        | Item<br>Code             | Item                                 | Item<br>Code                | Item                                 | Real/Mag. or<br>Imag./Phase |
|                        |                          |                                      | 13                          | Normal x at C                        | IP                          |
|                        |                          |                                      | 14                          | Shear xy at C                        | IP                          |
|                        |                          |                                      | 15                          | Shear zx at C                        | IP                          |
|                        | 13-18                    | Repeat items 7-12 for output point D | 16-24                       | Repeat items 7-15 for output point D |                             |
|                        | 19-24                    | Repeat items 7-12 for output point E | 25-33                       | Repeat items 7-15 for output point E |                             |
|                        | 25-30                    | Repeat items 7-12 for output point F | 34-42                       | Repeat items 7-15 for output point F |                             |
|                        | 31                       | Max longitudinal                     |                             |                                      |                             |
|                        | 32                       | Min longitudinal                     |                             |                                      |                             |
|                        | 33-60                    | Repeat items 5-32 for end 2          | 43-80                       | Repeat items 5-42 for end 2          |                             |

## Element Force Item Codes

All items are element forces (or moments) unless otherwise indicated.

Table 7-6 Element Force Item Codes Part 1

| Element Name<br>Code                       | Real Element Forces |                         | Complex Element Forces |                         | Real/Mag. or<br>Imag/Phase |
|--|---------------------|-------------------------|------------------------|-------------------------|----------------------------|
|  | Item<br>Code        | Item                    | Item<br>Code           | Item                    |                            |
| CBAR (34)<br><br>Linear                    | 2 <sup>1</sup>      | Bending End A plane 1   | 2 <sup>1</sup>         | Bending End A plane 1   | RM                         |
|  | 3 <sup>1</sup>      | Bending End A plane 2   | 3 <sup>1</sup>         | Bending End A plane 2   | RM                         |
|  | 4 <sup>1</sup>      | Bending End B plane 1   | 4 <sup>1</sup>         | Bending End B plane 1   | RM                         |
|  | 5 <sup>1</sup>      | Bending End B plane 2   | 5 <sup>1</sup>         | Bending End B plane 2   | RM                         |
|  | 6 <sup>1</sup>      | Shear plane 1           | 6 <sup>1</sup>         | Shear plane 1           | RM                         |
|  | 7 <sup>1</sup>      | Shear plane 2           | 7 <sup>1</sup>         | Shear plane 2           | RM                         |
|  | 8 <sup>1</sup>      | Axial force             | 8 <sup>1</sup>         | Axial force             | RM                         |
|  | 9 <sup>1</sup>      | Torque                  | 9 <sup>1</sup>         | Torque                  | RM                         |
|  |                     |                         | 10 <sup>1</sup>        | Bending End A plane 1   | IP                         |
|  |                     |                         | 11 <sup>1</sup>        | Bending End A plane 2   | IP                         |
|  |                     |                         | 12 <sup>1</sup>        | Bending End B plane 1   | IP                         |
|  |                     |                         | 13 <sup>1</sup>        | Bending End B plane 2   | IP                         |
|  |                     |                         | 14 <sup>1</sup>        | Shear plane 1           | IP                         |
|  |                     |                         | 15 <sup>1</sup>        | Shear plane 2           | IP                         |
|  |                     |                         | 16 <sup>1</sup>        | Axial force             | IP                         |
|  |                     |                         | 17 <sup>1</sup>        | Torque                  | IP                         |
| CBAR (100)<br><br>Intermediate<br>Stations | 2                   | Station Distance/Length | 2                      | Station Distance/Length |                            |
|  | 3                   | Bending Moment Plane 1  | 3                      | Bending Moment Plane 1  | RM                         |
|  | 4                   | Bending Moment Plane 2  | 4                      | Bending Moment Plane 2  | RM                         |
|  | 5                   | Shear Force Plane 1     | 5                      | Shear Force Plane 1     | RM                         |
|  | 6                   | Shear Force Plane 2     | 6                      | Shear Force Plane 2     | RM                         |
|  | 7                   | Axial                   | 7                      | Axial                   | RM                         |
|  | 8                   | Torque                  | 8                      | Torque                  | RM                         |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name Code | Real Element Forces |  | Complex Element Forces |   |                         |
|-------------------|---------------------|--|------------------------|---|-------------------------|
|                   | Item Code           | Item   | Item Code              | Item  | Real/Mag. or Imag/Phase |
|                   |                     | Item codes are given for end A. Addition of the quantity (K-1) * 8 to the item code points to the same information for other stations, where K is the station number. K=8 for end B and 2 through 7 for intermediate stations.   | 9                      | Bending Moment Plane 1  | IP                      |
|                   |                     |  | 10                     | Bending Moment Plane 2  | IP                      |
|                   |                     |  | 11                     | Shear Force Plane 1   | IP                      |
|                   |                     |  | 12                     | Shear Force Plane 2   | IP                      |
|                   |                     |  | 13                     | Axial   | IP                      |
|                   |                     |  | 14                     | Torque  | IP                      |
|                   |                     |  |                        | (Item codes above are given for End A. For codes 2 through 14 at intermediate stations add (K-1) * 13 where K is the station number, and for codes at End B, K+number of stations plus 1.)  |                         |
| CBEAM (2)         | 2                   | External grid point ID   | 2                      | External grid point ID  |                         |
| Linear            | 3                   | Station distance/length  | 3                      | Station distance/length   |                         |
|                   | 4 <sup>1</sup>      | Bending moment plane 1   | 4 <sup>1</sup>         | Bending moment plane 1  | RM                      |
|                   | 5 <sup>1</sup>      | Bending moment plane 2   | 5 <sup>1</sup>         | Bending moment plane 2  | RM                      |
|                   | 6 <sup>1</sup>      | Web shear plane 1  | 6 <sup>1</sup>         | Web shear plane 1   | RM                      |
|                   | 7 <sup>1</sup>      | Web shear plane 2  | 7 <sup>1</sup>         | Web shear plane 2   | RM                      |
|                   | 8 <sup>1</sup>      | Axial force  | 8 <sup>1</sup>         | Axial force   | RM                      |
|                   | 9 <sup>1</sup>      | Total torque   | 9 <sup>1</sup>         | Total torque  | RM                      |
|                   | 10 <sup>1</sup>     | Warping torque   | 10 <sup>1</sup>        | Warping torque  | RM                      |
|                   |                     | (Item codes are given for end A. Addition of the quantity (K-1) 9 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.) | 11 <sup>1</sup>        | Bending moment plane 1  | IP                      |
|                   |                     |  | 12 <sup>1</sup>        | Bending moment plane 2  | IP                      |
|                   |                     |  | 13 <sup>1</sup>        | Web shear plane 1   | IP                      |
|                   |                     |  | 14 <sup>1</sup>        | Web shear plane 2   | IP                      |
|                   |                     |  | 15 <sup>1</sup>        | Axial force   | IP                      |
|                   |                     |  | 16 <sup>1</sup>        | Total torque  | IP                      |
|                   |                     |  | 17 <sup>1</sup>        | Warping torque  | IP                      |
|                   |                     |  |                        | (Item codes are given for end A. Addition of the quantity (K-1) 16 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.) |                         |
| CBEAM3 (184)      | 2                   | External grid point ID   | 2                      | External grid point ID  |                         |
| Linear            | 3                   | Bending moment in y-direction  | 3                      | Bending moment in y-direction   | RM                      |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name<br>Code | Real Element Forces |  | Complex Element Forces |   | Real/Mag. or<br>Imag/Phase |
|----------------------|---------------------|--|------------------------|---|----------------------------|
|                      | Item<br>Code        | Item   | Item<br>Code           | Item  |                            |
|                      | 4                   | Bending moment in z-direction  | 4                      | Bending moment in z-direction   | RM                         |
|                      | 5                   | Shear force in y-direction   | 5                      | Shear force in y-direction  | RM                         |
|                      | 6                   | Shear force in z-direction   | 6                      | Shear force in z-direction  | RM                         |
|                      | 7                   | Axial force  | 7                      | Axial force   | RM                         |
|                      | 8                   | Total torque   | 8                      | Total torque  | RM                         |
|                      | 9                   | Bi-shear force   | 9                      | Bi-shear force  | RM                         |
|                      | 10                  | Bi-moment<br><br>(Item codes are given for end A.<br>They are repeated for end B and<br>mid-node C, respectively)      | 10                     | Bi-moment   | RM                         |
|                      |                     |  | 11                     | Bending moment in y-direction   | IP                         |
|                      |                     |  | 12                     | Bending moment in z-direction   | IP                         |
|                      |                     |  | 13                     | Shear force in y-direction  | IP                         |
|                      |                     |  | 14                     | Shear force in z-direction  | IP                         |
|                      |                     |  | 15                     | Axial force   | IP                         |
|                      |                     |  | 16                     | Total torque  | IP                         |
|                      |                     |  | 17                     | Bi-shear force  | IP                         |
|                      |                     |  | 18                     | Bi-moment<br><br>(Item codes are given for end A.<br>They are repeated for end B and<br>mid-node C, respectively) | IP                         |
| CBEND (69)           | 2                   | External grid point ID   | 2                      | External grid point ID  |                            |
|                      | 3 <sup>1</sup>      | Bending moment plane 1   | 3 <sup>1</sup>         | Bending moment plane 1  | RM                         |
|                      | 4 <sup>1</sup>      | Bending moment plane 2   | 4 <sup>1</sup>         | Bending moment plane 2  | RM                         |
|                      | 5 <sup>1</sup>      | Shear plane 1  | 5 <sup>1</sup>         | Shear plane 1   | RM                         |
|                      | 6 <sup>1</sup>      | Shear plane 2  | 6 <sup>1</sup>         | Shear plane 2   | RM                         |
|                      | 7 <sup>1</sup>      | Axial force  | 7 <sup>1</sup>         | Axial force   | RM                         |
|                      | 8 <sup>1</sup>      | Torque<br><br>(Item codes are given for end A.<br>Item codes 9 through 15 point to<br>the same information for end B.) | 8 <sup>1</sup>         | Torque  | RM                         |
|                      |                     |  | 9 <sup>1</sup>         | Bending moment plane 1  | IP                         |
|                      |                     |  | 10 <sup>1</sup>        | Bending moment plane 2  | IP                         |
|                      |                     |  | 11 <sup>1</sup>        | Shear plane 1   | IP                         |
|                      |                     |  | 12 <sup>1</sup>        | Shear plane 2   | IP                         |
|                      |                     |  | 13 <sup>1</sup>        | Axial force   | IP                         |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name Code             | Real Element Forces |                         | Complex Element Forces |   |                         |
|-------------------------------|---------------------|-------------------------|------------------------|---|-------------------------|
|                               | Item Code           | Item                    | Item Code              | Item  | Real/Mag. or Imag/Phase |
|                               |                     |                         | 14 <sup>1</sup>        | Torque<br><br>(Item codes are given for end A.<br>Item codes 15 through 27 point to<br>the same information for end B.) | IP                      |
| CBUSH (102)                   | 2 <sup>1</sup>      | Force-x                 | 2 <sup>1</sup>         | Force-x   | RM                      |
|                               | 3 <sup>1</sup>      | Force-y                 | 3 <sup>1</sup>         | Force-y   | RM                      |
|                               | 4 <sup>1</sup>      | Force-z                 | 4 <sup>1</sup>         | Force-z   | RM                      |
|                               | 5 <sup>1</sup>      | Moment-x                | 5 <sup>1</sup>         | Moment-x  | RM                      |
|                               | 6 <sup>1</sup>      | Moment-y                | 6 <sup>1</sup>         | Moment-y  | RM                      |
|                               | 7 <sup>1</sup>      | Moment-z                | 7 <sup>1</sup>         | Moment-z  | RM                      |
|                               |                     |                         | 8 <sup>1</sup>         | Force-x   | IP                      |
|                               |                     |                         | 9 <sup>1</sup>         | Force-y   | IP                      |
|                               |                     |                         | 10 <sup>1</sup>        | Force-z   | IP                      |
|                               |                     |                         | 11 <sup>1</sup>        | Moment-x  | IP                      |
|                               |                     |                         | 12 <sup>1</sup>        | Moment-y  | IP                      |
|                               |                     |                         | 13 <sup>1</sup>        | Moment-z  | IP                      |
| CCONEAX (35)                  | 2                   | Harmonic or point angle |                        | Not applicable  |                         |
|                               | 3                   | Bending moment v        |                        |   |                         |
|                               | 4                   | Bending moment u        |                        |   |                         |
|                               | 5                   | Twist moment            |                        |   |                         |
|                               | 6                   | Shear v                 |                        |   |                         |
|                               | 7                   | Shear u                 |                        |   |                         |
| CDAMP1 (20)                   |                     | Same as CELAS1          |                        | Same as CELAS1  |                         |
| CDAMP2 (21)                   |                     | Same as CELAS1          |                        | Same as CELAS1  |                         |
| CDAMP3 (22)                   |                     | Same as CELAS1          |                        | Same as CELAS1  |                         |
| CDAMP4 (23)                   |                     | Same as CELAS1          |                        | Same as CELAS1  |                         |
| CDUM3 thru<br>CDUM9 (55 - 61) | 2 <sup>1</sup>      | F1                      | 2 <sup>1</sup>         | F1  | RM                      |
|                               | 3 <sup>1</sup>      | F2                      | 3 <sup>1</sup>         | F2  | RM                      |
|                               | 4 <sup>1</sup>      | F3                      | 4 <sup>1</sup>         | F3  | RM                      |
|                               | 5 <sup>1</sup>      | F4                      | 5 <sup>1</sup>         | F4  | RM                      |
|                               | 6 <sup>1</sup>      | F5                      | 6 <sup>1</sup>         | F5  | RM                      |
|                               | 7 <sup>1</sup>      | F6                      | 7 <sup>1</sup>         | F6  | RM                      |
|                               | 8 <sup>1</sup>      | F7                      | 8 <sup>1</sup>         | F7  | RM                      |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name<br>Code | Real Element Forces |                   | Complex Element Forces |                   | Real/Mag. or<br>Imag/Phase |
|----------------------|---------------------|-------------------|------------------------|-------------------|----------------------------|
|                      | Item<br>Code        | Item              | Item<br>Code           | Item              |                            |
|                      | 9 <sup>1</sup>      | F8                | 9 <sup>1</sup>         | F8                | RM                         |
|                      | 10 <sup>1</sup>     | F9                | 10 <sup>1</sup>        | F9                | RM                         |
|                      |                     |                   | 11 <sup>1</sup>        | F1                | IP                         |
|                      |                     |                   | 12 <sup>1</sup>        | F2                | IP                         |
|                      |                     |                   | 13 <sup>1</sup>        | F3                | IP                         |
|                      |                     |                   | 14 <sup>1</sup>        | F4                | IP                         |
|                      |                     |                   | 15 <sup>1</sup>        | F5                | IP                         |
|                      |                     |                   | 16 <sup>1</sup>        | F6                | IP                         |
|                      |                     |                   | 17 <sup>1</sup>        | F7                | IP                         |
|                      |                     |                   | 18 <sup>1</sup>        | F8                | IP                         |
|                      |                     |                   | 19 <sup>1</sup>        | F9                | IP                         |
| CELAS1 (11)          | 2 <sup>1</sup>      | Force             | 2 <sup>1</sup>         | Force             | RM                         |
|                      | 3                   | Force             | 3                      | Force             | IP                         |
| CELAS2 (12)          |                     | Same as CELAS1    |                        | Same as CELAS1    |                            |
| CELAS3 (13)          |                     | Same as CELAS1    |                        | Same as CELAS1    |                            |
| CELAS4 (14)          |                     | Same as CELAS1    |                        | Same as CELAS1    |                            |
| CGAP (38)            | 2                   | Normal x          |                        | Not applicable    |                            |
|                      | 3                   | Shear y           |                        |                   |                            |
|                      | 4                   | Shear z           |                        |                   |                            |
|                      | 5                   | Axial u           |                        |                   |                            |
|                      | 6                   | Shear v           |                        |                   |                            |
|                      | 7                   | Shear w           |                        |                   |                            |
|                      | 8                   | Slip v            |                        |                   |                            |
|                      | 9                   | Slip w            |                        |                   |                            |
| CONROD (10)          | 2 <sup>1</sup>      | Axial force       | 2 <sup>1</sup>         | Axial force       | RM                         |
|                      | 3 <sup>1</sup>      | Torque            | 3 <sup>1</sup>         | Axial force       | IP                         |
|                      |                     |                   | 4 <sup>1</sup>         | Torque            | RM                         |
|                      |                     |                   | 5 <sup>1</sup>         | Torque            | IP                         |
| CQUAD4 (33)          | 2 <sup>1</sup>      | Membrane force x  | 2 <sup>1</sup>         | Membrane force x  | RM                         |
| Linear               | 3 <sup>1</sup>      | Membrane force y  | 3 <sup>1</sup>         | Membrane force y  | RM                         |
|                      | 4 <sup>1</sup>      | Membrane force xy | 4 <sup>1</sup>         | Membrane force xy | RM                         |
|                      | 5 <sup>1</sup>      | Bending moment x  | 5 <sup>1</sup>         | Bending moment x  | RM                         |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name Code | Real Element Forces |   | Complex Element Forces |                   | Real/Mag. or Imag/Phase |
|-------------------|---------------------|---|------------------------|-------------------|-------------------------|
|                   | Item Code           | Item  | Item Code              | Item              |                         |
|                   | 6 <sup>1</sup>      | Bending moment y  | 6 <sup>1</sup>         | Bending moment y  | RM                      |
|                   | 7 <sup>1</sup>      | Bending moment xy   | 7 <sup>1</sup>         | Bending moment xy | RM                      |
|                   | 8 <sup>1</sup>      | Shear x   | 8 <sup>1</sup>         | Shear x           | RM                      |
|                   | 9 <sup>1</sup>      | Shear y   | 9 <sup>1</sup>         | Shear y           | RM                      |
|                   |                     |   | 10 <sup>1</sup>        | Membrane force x  | IP                      |
|                   |                     |   | 11 <sup>1</sup>        | Membrane force y  | IP                      |
|                   |                     |   | 12 <sup>1</sup>        | Membrane force xy | IP                      |
|                   |                     |   | 13 <sup>1</sup>        | Bending moment x  | IP                      |
|                   |                     |   | 14 <sup>1</sup>        | Bending moment y  | IP                      |
|                   |                     |   | 15 <sup>1</sup>        | Bending moment xy | IP                      |
|                   |                     |   | 16 <sup>1</sup>        | Shear x           | IP                      |
|                   |                     |   | 17 <sup>1</sup>        | Shear y           | IP                      |
| CQUAD4 (95)       | 2-3                 | Theory or blank   |                        | Not applicable    |                         |
| Composite         | 4                   | Lamina number   |                        |                   |                         |
|                   | 5                   | FP (failure index)<br>/SP (strength ratio)<br>for direct stresses                 |                        |                   |                         |
|                   | 6                   | Failure mode for  |                        |                   |                         |
|                   |                     | Maximum strain theory   |                        |                   |                         |
|                   | 7                   | FB (failure index)<br>/SB (strength ratio) or -1 for<br>interlaminar shear-stress |                        |                   |                         |
|                   | 8                   | MAX of FP, FB or -1<br>or<br>MIN of SP, SB or -1                                  |                        |                   |                         |
|                   | 9                   | Failure flag  |                        |                   |                         |
| CQUAD4 (144)      | 1                   | EID   | 1                      | EID               |                         |
| Corner Output     | 2                   | CEN/  | 2                      | CEN/              |                         |
|                   | 3                   | 4   | 3                      | 4                 |                         |
|                   | 4                   | Membrane x  | 4                      | Membrane x        | RM                      |
|                   | 5                   | Membrane y  | 5                      | Membrane y        | RM                      |
|                   | 6                   | Membrane xy   | 6                      | Membrane xy       | RM                      |
|                   | 7                   | Bending x   | 7                      | Bending x         | RM                      |
|                   | 8                   | Bending y   | 8                      | Bending y         | RM                      |
|                   | 9                   | Bending xy  | 9                      | Bending xy        | RM                      |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name<br>Code | Real Element Forces |                                   | Complex Element Forces |                                   | Real/Mag. or<br>Imag/Phase |
|----------------------|---------------------|-----------------------------------|------------------------|-----------------------------------|----------------------------|
|                      | Item<br>Code        | Item                              | Item<br>Code           | Item                              |                            |
|                      | 10                  | Shear x                           | 10                     | Shear x                           | RM                         |
|                      | 11                  | Shear y                           | 11                     | Shear y                           | RM                         |
|                      | 12                  | Grid 1                            | 12                     | Membrane x                        | IP                         |
|                      | 13-20               | Same as 4 through 11 for corner 1 | 13                     | Membrane y                        | IP                         |
|                      | 21                  | Grid 2                            | 14                     | Membrane xy                       | IP                         |
|                      | 22-29               | Same as 4 through 11 for corner 2 | 15                     | Bending x                         | IP                         |
|                      | 30                  | Grid 3                            | 16                     | Bending y                         | IP                         |
|                      | 31-38               | Same as 4 through 11 for corner 3 | 17                     | Bending xy                        | IP                         |
|                      | 39                  | Grid 4                            | 18                     | Shear x                           | IP                         |
|                      | 40-47               | Same as 4 through 11 for corner 4 | 19                     | Shear y                           | IP                         |
|                      |                     |                                   | 20                     | Grid 1                            |                            |
|                      |                     |                                   | 21-36                  | Same as 4 through 19 for corner 1 |                            |
|                      |                     |                                   | 37                     | Grid 2                            |                            |
|                      |                     |                                   | 38-53                  | Same as 4 through 19 for corner 2 |                            |
|                      |                     |                                   | 54                     | Grid 3                            |                            |
|                      |                     |                                   | 55-70                  | Same as 4 through 19 for corner 3 |                            |
|                      |                     |                                   | 71                     | Grid 4                            |                            |
|                      |                     |                                   | 71-87                  | Same as 4 through 19 for corner 4 |                            |
| CQUAD8 (64)          | 1                   | EID                               | 1                      | EID                               |                            |
| Linear               | 2                   | CEN/                              | 2                      | CEN/                              |                            |
|                      | 3                   | 4                                 | 3                      | 4                                 |                            |
|                      | 4 <sup>1</sup>      | Membrane force x                  | 4 <sup>1</sup>         | Membrane force x                  | RM                         |
|                      | 5 <sup>1</sup>      | Membrane force y                  | 5 <sup>1</sup>         | Membrane force y                  | RM                         |
|                      | 6 <sup>1</sup>      | Membrane force xy                 | 6 <sup>1</sup>         | Membrane force xy                 | RM                         |
|                      | 7 <sup>1</sup>      | Bending moment x                  | 7 <sup>1</sup>         | Bending moment x                  | RM                         |
|                      | 8 <sup>1</sup>      | Bending moment y                  | 8 <sup>1</sup>         | Bending moment y                  | RM                         |
|                      | 9 <sup>1</sup>      | Bending moment xy                 | 9 <sup>1</sup>         | Bending moment xy                 | RM                         |
|                      | 10 <sup>1</sup>     | Shear x                           | 10 <sup>1</sup>        | Shear x                           | RM                         |

Table 7-6 Element Force Item Codes Part 1 (continued)

| Element Name Code                     | Real Element Forces |   | Complex Element Forces |   | Real/Mag. or Imag/Phase |
|---------------------------------------|---------------------|---|------------------------|---|-------------------------|
|                                       | Item Code           | Item                                    | Item Code              | Item                                    |                         |
|                                       | 11 <sup>1</sup>     | Shear y                                 | 11 <sup>1</sup>        | Shear y                                 | RM                      |
|                                       | 12                  | Grid1                                   | 12 <sup>1</sup>        | Membrane force x                        | IP                      |
|                                       | 13-20               | Same as items 4 through 11 for corner 1 | 13 <sup>1</sup>        | Membrane force y                        | IP                      |
|                                       | 21                  | Grid2                                   | 14 <sup>1</sup>        | Membrane force xy                       | IP                      |
|                                       | 22-29               | Same as items 4 through 11 for corner 2 | 15 <sup>1</sup>        | Bending moment x                        | IP                      |
|                                       | 30                  | Grid3                                   | 16 <sup>1</sup>        | Bending moment y                        | IP                      |
|                                       | 31-38               | Same as items 4 through 11 for corner 3 | 17 <sup>1</sup>        | Bending moment xy                       | IP                      |
|                                       | 39                  | Grid4                                   | 18 <sup>1</sup>        | Shear x                                 | IP                      |
|                                       | 40-47               | Same as items 4 through 11 for corner 4 | 19 <sup>1</sup>        | Shear y                                 | IP                      |
|                                       |                     |   | 20                     | Grid1                                   |                         |
|                                       |                     |   | 21-36                  | Same as items 4 through 19 for corner 1 |                         |
| 37                                    |                     |   | 37                     | Grid2                                   |                         |
|                                       |                     |   | 38-53                  | Same as items 4 through 19 for corner 2 |                         |
|                                       |                     |   | 54                     | Grid3                                   |                         |
|                                       |                     |   | 55-70                  | Same as items 4 through 19 for corner 3 |                         |
|                                       |                     |   | 71                     | Grid4                                   |                         |
|                                       |                     |   | 72-87                  | Same as items 4 through 19 for corner 4 |                         |
| CQUAD8 <sup>2</sup> (96)<br>Composite |                     | Same as CQUAD4(95)                      |                        | Same as CQUAD4(95)                      |                         |
| CQUADR (82)                           |                     | Same as CQUAD8(64)                      |                        | Same as CQUAD8(64)                      |                         |
| CQUADR (235)<br>Center                |                     | Same as CQUAD4(33)                      |                        | Same as CQUAD4(33)                      |                         |

Table 7-7 Element Force Item Codes Part 2

| Element Name<br>Code | Real Element Forces |                    | Complex Element Forces |                    | Real/Mag. or<br>Imag/Phase |
|----------------------|---------------------|--------------------|------------------------|--------------------|----------------------------|
|                      | Item<br>Code        | Item               | Item<br>Code           | Item               |                            |
| CROD (1)             |                     | Same as CONROD(10) |                        | Same as CONROD(10) |                            |
| CSHEAR (4)           | 2 <sup>1</sup>      | Force 4 to 1       | 2 <sup>1</sup>         | Force 4 to 1       | RM                         |
|                      | 3 <sup>1</sup>      | Force 2 to 1       | 3 <sup>1</sup>         | Force 2 to 1       | RM                         |
|                      | 4 <sup>1</sup>      | Force 1 to 2       | 4 <sup>1</sup>         | Force 1 to 2       | RM                         |
|                      | 5 <sup>1</sup>      | Force 3 to 2       | 5 <sup>1</sup>         | Force 3 to 2       | RM                         |
|                      | 6 <sup>1</sup>      | Force 2 to 3       | 6 <sup>1</sup>         | Force 2 to 3       | RM                         |
|                      | 7 <sup>1</sup>      | Force 4 to 3       | 7 <sup>1</sup>         | Force 4 to 3       | RM                         |
|                      | 8 <sup>1</sup>      | Force 3 to 4       | 8 <sup>1</sup>         | Force 3 to 4       | RM                         |
|                      | 9 <sup>1</sup>      | Force 1 to 4       | 9 <sup>1</sup>         | Force 1 to 4       | RM                         |
|                      | 10 <sup>1</sup>     | Kick force on 1    | 10 <sup>1</sup>        | Force 4 to 1       | IP                         |
|                      | 11 <sup>1</sup>     | Shear 12           | 11 <sup>1</sup>        | Force 2 to 1       | IP                         |
|                      | 12 <sup>1</sup>     | Kick force on 2    | 12 <sup>1</sup>        | Force 1 to 2       | IP                         |
|                      | 13 <sup>1</sup>     | Shear 23           | 13 <sup>1</sup>        | Force 3 to 2       | IP                         |
|                      | 14 <sup>1</sup>     | Kick force on 3    | 14 <sup>1</sup>        | Force 2 to 3       | IP                         |
|                      | 15 <sup>1</sup>     | Shear 34           | 15 <sup>1</sup>        | Force 4 to 3       | IP                         |
|                      | 16 <sup>1</sup>     | Kick force on 4    | 16 <sup>1</sup>        | Force 3 to 4       | IP                         |
|                      | 17 <sup>1</sup>     | Shear 41           | 17 <sup>1</sup>        | Force 1 to 4       | IP                         |
|                      |                     |                    | 18 <sup>1</sup>        | Kick force on 1    | RM                         |
|                      |                     |                    | 19 <sup>1</sup>        | Shear 12           | RM                         |
|                      |                     |                    | 20 <sup>1</sup>        | Kick force on 2    | RM                         |
|                      |                     |                    | 21 <sup>1</sup>        | Shear 23           | RM                         |
|                      |                     |                    | 22 <sup>1</sup>        | Kick force on 3    | RM                         |
|                      |                     |                    | 23 <sup>1</sup>        | Shear 34           | RM                         |
|                      |                     |                    | 24 <sup>1</sup>        | Kick force on 4    | RM                         |
|                      |                     |                    | 25 <sup>1</sup>        | Shear 41           | RM                         |
|                      |                     |                    | 26 <sup>1</sup>        | Kick force on 1    | IP                         |
|                      |                     |                    | 27 <sup>1</sup>        | Shear 12           | IP                         |
|                      |                     |                    | 28 <sup>1</sup>        | Kick force on 2    | IP                         |
|                      |                     |                    | 29 <sup>1</sup>        | Shear 23           | IP                         |
|                      |                     |                    | 30 <sup>1</sup>        | Kick force on 3    | IP                         |
|                      |                     |                    | 31 <sup>1</sup>        | Shear 34           | IP                         |

Table 7-7 Element Force Item Codes Part 2 (continued)

| Element Name<br>Code                  | Real Element Forces |   | Complex Element Forces |   | Real/Mag. or<br>Imag/Phase |
|---------------------------------------|---------------------|---|------------------------|---|----------------------------|
|                                       | Item<br>Code        | Item                                    | Item<br>Code           | Item                                    |                            |
|                                       |                     |   | 32 <sup>1</sup>        | Kick force on 4                         | IP                         |
|                                       |                     |   | 33 <sup>1</sup>        | Shear 41                                | IP                         |
| CTRIA3 (74)<br>Linear                 |                     | Same as CQUAD4(33)                      |                        | Same as CQUAD4(33)                      |                            |
| CTRIA3 <sup>2</sup> (97)<br>Composite |                     | Same as CQUAD4(95)                      |                        | Same as CQUAD4(95)                      |                            |
| CTRIA6 (75)<br>Linear                 | 4 <sup>1</sup>      | Membrane force x                        | 4 <sup>1</sup>         | Membrane force x                        | RM                         |
|                                       | 5 <sup>1</sup>      | Membrane force y                        | 5 <sup>1</sup>         | Membrane force y                        | RM                         |
|                                       | 6 <sup>1</sup>      | Membrane force xy                       | 6 <sup>1</sup>         | Membrane force xy                       | RM                         |
|                                       | 7 <sup>1</sup>      | Bending moment x                        | 7 <sup>1</sup>         | Bending moment x                        | RM                         |
|                                       | 8 <sup>1</sup>      | Bending moment y                        | 8 <sup>1</sup>         | Bending moment y                        | RM                         |
|                                       | 9 <sup>1</sup>      | Bending moment xy                       | 9 <sup>1</sup>         | Bending moment xy                       | RM                         |
|                                       | 10 <sup>1</sup>     | Shear x                                 | 10 <sup>1</sup>        | Shear x                                 | RM                         |
|                                       | 11 <sup>1</sup>     | Shear y                                 | 11 <sup>1</sup>        | Shear y                                 | RM                         |
|                                       | 13-20               | Same as items 4 through 11 for corner 1 | 12 <sup>1</sup>        | Membrane force x                        | IP                         |
|                                       | 22-29               | Same as items 4 through 11 for corner 2 | 13 <sup>1</sup>        | Membrane force y                        | IP                         |
|                                       | 31-38               | Same as items 4 through 11 for corner 3 | 14 <sup>1</sup>        | Membrane force xy                       | IP                         |
|                                       |                     |   | 15 <sup>1</sup>        | Bending moment x                        | IP                         |
|                                       |                     |   | 16 <sup>1</sup>        | Bending moment y                        | IP                         |
|                                       |                     |   | 17 <sup>1</sup>        | Bending moment xy                       | IP                         |
|                                       |                     |   | 18 <sup>1</sup>        | Shear x                                 | IP                         |
|                                       |                     |   | 19 <sup>1</sup>        | Shear y                                 | IP                         |
|                                       |                     |   | 21-36                  | Same as items 4 through 19 for corner 1 |                            |
|                                       |                     |   | 38-53                  | Same as items 4 through 19 for corner 2 |                            |
|                                       |                     |   | 55-70                  | Same as items 4 through 19 for corner 3 |                            |
| CTRIA6 <sup>2</sup> (98)<br>Composite |                     | Same as CQUAD4(95)                      |                        | Same as CQUAD4(95)                      |                            |
| CTRIAR (70)                           |                     | Same as CTRIA6(75)                      |                        | Same as CTRIA6(75)                      |                            |

Table 7-7 Element Force Item Codes Part 2 (continued)

| Element Name<br>Code   | Real Element Forces |                                     | Complex Element Forces |                          | Real/Mag. or<br>Imag/Phase |
|--|---------------------|-------------------------------------|------------------------|--------------------------|----------------------------|
|  | Item<br>Code        | Item                                | Item<br>Code           | Item                     |                            |
| <b>CTRIAR (236)</b><br>Center  | Same as CQUAD4(33)  |                                     | Same as CQUAD4(33)     |                          |                            |
| <b>CTUBE (3)</b>   | Same as CONROD(10)  |                                     | Same as CONROD(10)     |                          |                            |
| <b>CVISC (24)</b>  | Not applicable      |                                     | Same as CONROD(10)     |                          |                            |
| <b>CWELDP (118)</b><br>if PARTPAT or<br>ELPAT  | 2                   | mz bending end A plane 1            | 2                      | mz bending end A plane 1 | RM                         |
|  | 3                   | my bending end A plane 2            | 3                      | my bending end A plane 2 | RM                         |
|  | 4                   | mz bending end B plane 1            | 4                      | mz bending end B plane 1 | RM                         |
|  | 5                   | my bending end B plane 2            | 5                      | my bending end B plane 2 | RM                         |
|  | 6                   | fy shear force plane 1              | 6                      | fy shear force plane 1   | RM                         |
|  | 7                   | fz shear force plane 2              | 7                      | fz shear force plane 2   | RM                         |
|  | 8                   | fx axial force                      | 8                      | fx axial force           | RM                         |
| <b>CWELDC (117)</b><br>if MSET = OFF   | 9                   | mx torque                           | 9                      | mx torque                | RM                         |
|  |                     |                                     | 10                     | mz bending end A plane 1 | IP                         |
|  |                     |                                     | 11                     | my bending end A plane 2 | IP                         |
| <b>CWELD (200)</b><br>if MSET = ON   |                     |                                     | 12                     | mz bending end B plane 1 | IP                         |
|  |                     |                                     | 13                     | my bending end B plane 2 | IP                         |
|  |                     |                                     | 14                     | fy shear force plane 1   | IP                         |
|  |                     |                                     | 15                     | fz shear force plane 2   | IP                         |
|  |                     |                                     | 16                     | fx axial force           | IP                         |
|  |                     |                                     | 17                     | mx torque                | IP                         |
| <b>VUQUAD (189)</b><br><b>VUTRIA (190)</b><br>for QUADp and<br>TRIAp if 4th char.<br>of ICORD=X Y, Z<br>(local coordinate<br>system) | 1                   | VU element ID * 10 +<br>device code |                        |                          |                            |
|  | 2                   | Parent p-element ID                 |                        |                          |                            |
|  | 3                   | CID coordinate system ID            |                        |                          |                            |
|  | 4                   | ICORD flat/curved etc.              |                        |                          |                            |
|  | 5                   | THETA material angle                |                        |                          |                            |
|  | 6                   | 0.0                                 |                        |                          |                            |
|  | 7                   | VU grid ID for corner 1             |                        |                          |                            |
|  | 8                   | Membrane Force x                    | 8                      | Membrane Force x         | RM                         |
|  | 9                   | Membrane Force y                    | 9                      | Membrane Force y         | RM                         |
|  | 10                  | Membrane Force xy                   | 10                     | Membrane Force xy        | RM                         |
|  | 11                  | 0.0                                 | 11                     | 0.0                      | RM                         |

Table 7-7 Element Force Item Codes Part 2 (continued)

| Element Name Code  | Real Element Forces |   | Complex Element Forces |   | Real/Mag. or Imag/Phase |
|--|---------------------|---|------------------------|---|-------------------------|
|  | Item Code           | Item                                    | Item Code              | Item                                    |                         |
|  | 12                  | 0.0                                     | 12                     | 0.0                                     | RM                      |
|  | 13                  | 0.0                                     | 13                     | 0.0                                     | RM                      |
|  | 14                  | Bending Moment x                        | 14                     | Bending Moment x                        | RM                      |
|  | 15                  | Bending Moment y                        | 15                     | Bending Moment y                        | RM                      |
|  | 16                  | Bending Moment xy                       | 16                     | Bending Moment xy                       | RM                      |
|  | 17                  | Shear zx                                | 17                     | Shear zx                                | RM                      |
|  | 18                  | Shear yz                                | 18                     | Shear yz                                | RM                      |
|  | 19                  | 0.0                                     | 19                     | 0.0                                     | RM                      |
|  |                     |   | 20                     | Membrane Force x                        | IP                      |
|  |                     |   | 21                     | Membrane Force y                        | IP                      |
|  |                     |   | 22                     | Membrane Force xy                       | IP                      |
|  |                     |   | 23                     | 0.0                                     | IP                      |
|  |                     |   | 24                     | 0.0                                     | IP                      |
|  |                     |   | 25                     | 0.0                                     | IP                      |
|  |                     |   | 26                     | Bending Moment x                        | IP                      |
|  |                     |   | 27                     | Bending Moment y                        | IP                      |
|  |                     |   | 28                     | Bending Moment xy                       | IP                      |
|  |                     |   | 29                     | Shear zx                                | IP                      |
|  |                     |   | 30                     | Shear yz                                | IP                      |
|  |                     |   | 31                     | 0.0                                     | IP                      |
|  | 20-32               | Repeat items 7-19 for corner 2          | 32-56                  | Repeat items 7-31 for corner 2          |                         |
|  | 33-45               | Repeat items 7-19 for corner 3          | 57-81                  | Repeat items 7-31 for corner 3          |                         |
|  | 46-58               | Repeat items 7-19 for corner 4 (VUQUAD) | 82-106                 | Repeat items 7-31 for corner 4 (VUQUAD) |                         |
| <b>VUQUAD (189)<br/>VUTRIA (190)</b><br><br>for QUADp and<br>TRIAP if 4th char.<br>of ICORD=F (fixed<br>coordinate system) | 1                   | VU element ID *10 + device code         |                        |   |                         |
|  | 2                   | Parent p-element ID                     |                        |   |                         |
|  | 3                   | CID coordinate system ID                |                        |   |                         |
|  | 4                   | ICORD flat/curved etc.                  |                        |   |                         |
|  | 5                   | THETA material angle                    |                        |   |                         |
|  | 6                   | 0.0                                     |                        |   |                         |
|  | 7                   | VU grid ID for corner 1                 |                        |   |                         |

Table 7-7 Element Force Item Codes Part 2 (continued)

| Element Name<br>Code      | Real Element Forces |   | Complex Element Forces |   | Real/Mag. or<br>Imag/Phase |
|---------------------------|---------------------|---|------------------------|---|----------------------------|
|                           | Item<br>Code        | Item                                    | Item<br>Code           | Item                                    |                            |
|                           | 8                   | Membrane Force x                        | 8                      | Membrane Force x                        | RM                         |
|                           | 9                   | Membrane Force y                        | 9                      | Membrane Force y                        | RM                         |
|                           | 10                  | Membrane Force xy                       | 10                     | Membrane Force xy                       | RM                         |
|                           | 11                  | Membrane Force yz                       | 11                     | Membrane Force yz                       | RM                         |
|                           | 12                  | Membrane Force zx                       | 12                     | Membrane Force zx                       | RM                         |
|                           | 13                  | Membrane Force z                        | 13                     | Membrane Force z                        | RM                         |
|                           | 14                  | Bending Moment x                        | 14                     | Bending Moment x                        | RM                         |
|                           | 15                  | Bending Moment y                        | 15                     | Bending Moment y                        | RM                         |
|                           | 16                  | Bending Moment xy                       | 16                     | Bending Moment xy                       | RM                         |
|                           | 17                  | Bending Moment yz                       | 17                     | Bending Moment yz                       | RM                         |
|                           | 18                  | Bending Moment zx                       | 18                     | Bending Moment zx                       | RM                         |
|                           | 19                  | Bending Moment z                        | 19                     | Bending Moment z                        | RM                         |
|                           |                     |   | 20                     | Membrane Force x                        | IP                         |
|                           |                     |   | 21                     | Membrane Force y                        | IP                         |
|                           |                     |   | 22                     | Membrane Force xy                       | IP                         |
|                           |                     |   | 23                     | Membrane Force yz                       | IP                         |
|                           |                     |   | 24                     | Membrane Force zx                       | IP                         |
|                           |                     |   | 25                     | Membrane Force z                        | IP                         |
|                           |                     |   | 26                     | Bending Moment x                        | IP                         |
|                           |                     |   | 27                     | Bending Moment y                        | IP                         |
|                           |                     |   | 28                     | Bending Moment xy                       | IP                         |
|                           |                     |   | 29                     | Bending Moment yz                       | IP                         |
|                           |                     |   | 30                     | Bending Moment zx                       | IP                         |
|                           |                     |   | 31                     | Bending Moment z                        | IP                         |
|                           | 20-32               | Repeat items 7-19 for corner 2          | 32-56                  | Repeat items 7-31 for corner 2          |                            |
|                           | 33-45               | Repeat items 7-19 for corner 3          | 57-81                  | Repeat items 7-31 for corner 3          |                            |
|                           | 46-58               | Repeat items 7-19 for corner 4 (VUQUAD) | 82-106                 | Repeat items 7-31 for corner 4 (VUQUAD) |                            |
| VUBEAM (191)<br>for BEAMP | 1                   | VU element ID * 10 + device code        |                        |   |                            |
|                           | 2                   | Parent p-element ID                     |                        |   |                            |
|                           | 3                   | CID coordinate system ID                |                        |   |                            |

Table 7-7 Element Force Item Codes Part 2 (continued)

| Element Name Code | Real Element Forces |                             | Complex Element Forces |                             | Real/Mag. or Imag/Phase |
|-------------------|---------------------|-----------------------------|------------------------|-----------------------------|-------------------------|
|                   | Item Code           | Item                        | Item Code              | Item                        |                         |
|                   | 4                   | ICORD (not used)            |                        |                             |                         |
|                   | 5                   | VU grid ID for end 1        |                        |                             |                         |
|                   | 6                   | x/L position                |                        |                             |                         |
|                   | 7                   | Force x                     | 7                      | Force x                     | RM                      |
|                   | 8                   | Shear Force y               | 8                      | Shear Force y               | RM                      |
|                   | 9                   | Shear Force z               | 9                      | Shear Force z               | RM                      |
|                   | 10                  | Torsional Moment x          | 10                     | Torsional Moment x          | RM                      |
|                   | 11                  | Bending Moment y            | 11                     | Bending Moment y            | RM                      |
|                   | 12                  | Bending Moment z            | 12                     | Bending Moment z            | RM                      |
|                   |                     |                             | 13                     | Force x                     | IP                      |
|                   |                     |                             | 14                     | Shear Force y               | IP                      |
|                   |                     |                             | 15                     | Shear Force z               | IP                      |
|                   |                     |                             | 16                     | Torsional Moment x          | IP                      |
|                   |                     |                             | 17                     | Bending Moment y            | IP                      |
|                   |                     |                             | 18                     | Bending Moment z            | IP                      |
|                   | 13-20               | Repeat items 5-12 for end 2 | 19-32                  | Repeat items 5-18 for end 2 |                         |

## Fluid Virtual Mass Pressure Item Codes

Table 7-8 Fluid Virtual Mass Pressure Item Codes

| Element Name | Real Fluid Pressure |                | Complex Fluid Pressure |          |                          |
|--------------|---------------------|----------------|------------------------|----------|--------------------------|
|              | Code                | Item           | Code                   | Item     | Real/Mag. or Imag./Phase |
| Plate        | 2                   | Fluid pressure | 2                      | Pressure | RM                       |
| Family       |                     |                | 3                      | Pressure | IP                       |

Table 7-9 Heat Transfer Item Codes (Curve type is FLUX.)

| Element Name (Code)    | Code           | Item              |
|------------------------|----------------|-------------------|
| Heat Transfer Elements | 2              | Element type      |
|                        | 3 <sup>1</sup> |                   |
|                        | 4 <sup>1</sup> | x gradient        |
|                        | 5 <sup>1</sup> | y gradient        |
|                        | 6 <sup>1</sup> | z gradient        |
|                        | 7              | x flux            |
|                        | 8              | y flux            |
|                        | 9              | z flux            |
| CHBDYE                 | 4              | Applied load      |
| (107)                  | 5              | Free convection   |
|                        | 6              | Forced convection |
|                        | 7              | Radiation         |
|                        | 8              | Total             |
| CHBDYG                 | Same as        | Same as CHBDYE    |
| (108)                  | CHBDYE         |                   |
| CHBDYP                 | Same as        | Same as CHBDYE    |
| (109)                  | CHBDYE         |                   |

## 2D Slideline and 3D Surface Contact Item Codes

Table 7-10 Contact Item Codes

| Element Name<br>(Code) | Real Element Data |   |                           |
|------------------------|-------------------|---|---------------------------|
|                        | Item<br>Code      | Slideline Item                          | 3D Surface Item (SOL 600) |
| <b>CSLIFID (116)</b>   | 1                 | Slave grid point                        | GRID ID                   |
|                        | 2                 | Contact region identification number    | (39) Contact Touched Body |
|                        | 3                 | Master grid 1                           |                           |
|                        | 4                 | Master grid 2                           |                           |
|                        | 5                 | Surface coordinate                      | (38) Contact Status       |
|                        | 6                 | Normal force                            | (35) Normal Force         |
|                        | 7                 | Shear force                             | (37) Friction Force       |
|                        | 8                 | Normal stress                           | (34) Normal Stress        |
|                        | 9                 | Shear stress                            | (36) Friction Stress      |
|                        | 10                | Normal gap                              |                           |
|                        | 11                | Slip                                    |                           |
|                        | 12                | Slip ratio (Shear force/u*normal force) |                           |
|                        | 13-14             | Slip code (Character)                   |                           |

1. Numbers in parenthesis refer to MARCOUT nodal post codes.
2. 3D Surface contact is available in SOL 600 only.

Table 7-11 Contact Item Code for Nodal Data (SOL 400 only)

| Element Code | Item | Real Nodal Data<br>3D Surface Item (SOL 400 only) |
|--------------|------|---|
| 203          | 1    | Grid point ID                                     |
|              | 2    | First touched body                                |
|              | 3    | Second touched body                               |
|              | 4    | Third touched body                                |
|              | 5    | Contact status (Remark 7)                         |
|              | 6    | Normal contact force magnitude                    |
|              | 7    | Friction contact force magnitude                  |
|              | 8    | Normal stress                                     |
|              | 9    | Friction stress 1                                 |

Table 7-11 Contact Item Code for Nodal Data (SOL 400 only)

| Element Code | Item | Real Nodal Data<br>3D Surface Item (SOL 400 only) |
|--------------|------|---|
|              | 10   | Friction stress 2                                 |
|              | 11   | Contact normal force component $F_{nx}$           |
|              | 12   | Contact normal force component $F_{ny}$           |
|              | 13   | Contact normal force component $F_{nz}$           |
|              | 14   | Contact friction force component $F_{fx}$         |
|              | 15   | Contact friction force component $F_{fy}$         |
|              | 16   | Contact friction force component $F_{fz}$         |

Table 7-12 Contact Item Code for Body Data (SOL 400 only)

| Element Code | Item | Rigid Body Data<br>3D Surface Item (SOL 400 Only) |
|--------------|------|---|
| 203          | 1    | BCBODY ID   |
|              | 2-17 | [XMAT] <sub>4x4</sub>                             |
|              | 18   | Contact force component $RF_x$                    |
|              | 19   | Contact force component $RF_y$                    |
|              | 20   | Contact force component $RF_z$                    |
|              | 21   | Contact moment component $RM_x$                   |
|              | 22   | Contact moment component $RM_y$                   |
|              | 23   | Contact moment component $RM_z$                   |

**Remarks:**

1.  $F_{nx}, F_{ny}, F_{nz}$  are normal force components in the global coordinate system.
2.  $F_{fx}, F_{fy}, F_{fz}$  are friction force components in the global coordinate system.
3. Normal force magnitude is square root of sum of squares of  $F_{nx}, F_{ny}, F_{nz}$ .
4. Friction force magnitude is square root of sum of squares of  $F_{fx}, F_{fy}, F_{fz}$  with sign of largest absolute of these values.
5. Contact touched body applies to slave grids (touching node) only.
6. A slave grid may not contact more than 3 bodies.
7. Only slave grids have non-zero contact status or touched bodies.
8.  $RF_x, RF_y, RF_z, RM_x, RM_y, RM_z$  are the contact force and moment in the global coordinate system that applied on the centroid of the corresponding rigid body.

9. [XMAT] is the translation matrix to compute the new location of the rigid body.
10. These contact outputs, both [Table 7-11](#) and [Table 7-12](#) are saved in datablock OFCON3D.

## Element Strain Energy Item Codes

Table 7-13 Element Strain Energy Item Codes

| Element Name           | Real Element Data |                               |
|------------------------|-------------------|-------------------------------|
|                        | Item Code         | Item                          |
| Element groups A and B | 2                 | Element strain energy         |
| Element groups A and B | 3                 | Percent of total energy       |
| Element group A        | 4                 | Element strain energy density |

Remark:

1. Element group A includes elements of CBAR, CBEAM, CBEND, CONROD, CHEXA, CPENTA, CPYRAM, CQUAD4, CQUADR, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, and CTUBE. Element group B includes elements of CELAS1, CELAS2, CELAS3, AND CGAP.

## Fatigue Item Codes

Note that the codes of the [Table 7-14](#) through [Table 7-17](#) are limited to those that can be referenced on a DRESP1 entry with RTYPE= FATGUE.

**Table 7-14** Fatigue Item Codes for LOC=ELEM on FTGPARM

| Element Name  | Item Code | Real Element Data Item  |
|---|-----------|---|
| Element Group A and<br>Element Group B<br>(for Z1 layer - bottom) | 4         | Fatigue life in <i>Repeats</i> of the loading sequence<br>See remark 4. |
|   | (-4)      |   |
|   | 5 (-5)    | Log of fatigue life in <i>Repeats</i> of the loading sequence           |
|   | 6 (-6)    | Fatigue life in user defined fatigue equivalent units                   |
|   | 7 (-7)    | Log of fatigue life in user defined fatigue equivalent units            |
|   | 8 (-8)    | Fatigue damage  |
|   | 9 (-9)    | Log of fatigue damage   |
| Element Group B<br>(for Z2 layer - top)                           | 12 (-12)  | Scale factor from Factor of Safety analysis                             |
|   | 18        | Fatigue life in <i>Repeats</i> of the loading sequence                  |
|   | 19        | Log of fatigue life in <i>Repeats</i> of the loading sequence           |
|   | 20        | Fatigue life in user defined fatigue equivalent units                   |
|   | 21        | Log of fatigue life in user defined fatigue equivalent units            |
|   | 22        | Fatigue damage  |
|   | 23        | Log of fatigue damage   |
|   | 26        | Scale factor from Factor of Safety analysis                             |

### Remark:

1. Element Group A consists of elements: CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CSHEAR
2. Element Group B consists of elements: CQUAD4, CQUAD8, CQUADR, CTRIA6, CTRIAR
3. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARM use the same item codes in [Table 7-14](#).
4. For Element Group B (shell elements), if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular set of elements or property sets.

Note: SOL 200 only supports NODE and not NODA on the FTGPARM entry

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM

| Element Name         | Item Codes                          | Real Element Data Item                                |
|----------------------|-------------------------------------|---|
| CTETRA               | 4,16,28,40                          | Life in <i>Repeats</i> of the loading sequence        |
|                      | 5,17,29,41                          | Log of life in <i>Repeats</i> of the loading sequence |
|                      | 6,18,30,42                          | Life in user defined fatigue equivalent units         |
|                      | 7,19,31,43                          | Log of life in user defined equivalent units          |
|                      | 8,20,32,44                          | Fatigue damage  |
|                      | 9,21,33,45                          | Log of fatigue damage                                 |
|                      | 12,24,36,48                         | Scale factor from Factor of Safety analysis           |
| CQUAD4               | 4,33,62,91                          | Life in <i>Repeats</i> of the loading sequence        |
|                      | (-4,-33,-62,-91)                    | See remark 8.   |
|                      | 5,34,63,92<br>(-5,-34,-63,-92)      | Log of life in <i>Repeats</i> of the loading sequence |
|                      | 6,35,64,93<br>(-6,-35,-64,-93)      | Life in user defined fatigue equivalent units         |
|                      | 7, 36, 65, 94<br>(-7,-36,-65, -94)  | Log of life in user defined equivalent units          |
|                      | 8, 37, 66, 95<br>(-8,-37,-66,-95)   | Fatigue damage  |
|                      | 9, 38, 67, 96<br>(-9,-38,-67-96)    | Log of fatigue damage                                 |
| (for Z2 layer - top) | 12, 41, 70, 99<br>(-12,-41,-70,-99) | Scale factor from Factor of Safety analysis           |
|                      | 18, 47, 76, 105                     | Life in <i>Repeats</i> of the loading sequence        |
|                      | 19, 48, 77, 106                     | Log of life in <i>Repeats</i> of the loading sequence |
|                      | 20, 49, 78, 107                     | Life in user defined fatigue equivalent units         |
|                      | 21, 50, 79, 108                     | Log of life in user defined equivalent units          |
|                      | 22, 51, 80, 109                     | Fatigue damage  |
|                      | 23, 52, 81, 110                     | Log of fatigue damage                                 |
|                      | 26, 55, 84, 113                     | Scale factor from Factor of Safety analysis           |

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

| Element Name                      | Item Codes                    | Real Element Data Item                                |
|-----------------------------------|-------------------------------|---|
| CTRIA6                            | 4, 33, 62                     | Life in <i>Repeats</i> of the loading sequence        |
| CTRIAR<br>(for Z1 layer - bottom) | (-4, -33, -62)                | See remark 8.   |
|                                   | 5, 34, 63<br>(-5, -34, -63)   | Log of life in <i>Repeats</i> of the loading sequence |
|                                   | 6, 35, 64<br>(-6, -35, -64)   | Life in user defined fatigue equivalent units         |
|                                   | 7, 36, 6<br>(-7, -36, -6)     | Log of life in user defined equivalent units          |
|                                   | 8, 37, 66<br>(-8, -37, -66)   | Fatigue damage  |
|                                   | 9, 38, 67<br>(-9, -38, -67)   | Log of fatigue damage                                 |
|                                   | 12, 41, 70<br>(-12, -41, -70) | Scale factor from Factor of Safety analysis           |
| (for Z2 layer - top)              | 18, 47, 76                    | Life in <i>Repeats</i> of the loading sequence        |
|                                   | 19, 48, 77                    | Log of life in <i>Repeats</i> of the loading sequence |
|                                   | 20, 50, 79                    | Life in user defined fatigue equivalent units         |
|                                   | 22, 51, 80                    | Log of life in user defined equivalent units          |
|                                   | 23, 52, 81                    | Fatigue damage  |
|                                   | 26, 55, 84                    | Log of fatigue damage                                 |
|                                   |                               | Scale factor from Factor of Safety analysis           |
| CPENTA                            | 4,16,28,40,52,64              | Life in <i>Repeats</i> of the loading sequence        |
|                                   | 5,17,29,41,53,65              | Log of life in <i>Repeats</i> of the loading sequence |
|                                   | 6,18,30,42,54,66              | Life in user defined fatigue equivalent units         |
|                                   | 7,19,31,43,55,67              | Log of life in user defined equivalent units          |
|                                   | 8,20,32,44,56,68              | Fatigue damage  |
|                                   | 9,21,33,45,57,69              | Log of fatigue damage                                 |
|                                   | 12,24,36,48,60,72             | Scale factor from Factor of Safety analysis           |

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

| Element Name | Item Codes              | Real Element Data Item                                |
|--------------|-------------------------|---|
| CPYRAM       | 4,16,28,40,52           | Life in <i>Repeats</i> of the loading sequence        |
|              | 5,17,29,41,53           | Log of life in <i>Repeats</i> of the loading sequence |
|              | 6,18,30,42,54           | Life in user defined fatigue equivalent units         |
|              | 7,19,31,43,55           | Log of life in user defined equivalent units          |
|              | 8,20,32,44,56           | Fatigue damage  |
|              | 9,21,33,45,57           | Log of fatigue damage                                 |
|              | 12,24,36,48,60          | Scale factor from Factor of Safety analysis           |
| CHEXA        | 4,16,28,40,52,64,76,88  | Life in <i>Repeats</i> of the loading sequence        |
|              | 5,17,29,41,53,65,77,89  | Log of life in <i>Repeats</i> of the loading sequence |
|              | 6,18,30,42,54,66,78,90  | Life in user defined fatigue equivalent units         |
|              | 7,19,31,43,55,67,79,91  | Log of life in user defined equivalent units          |
|              | 8,20,32,44,56,68,80,92  | Fatigue damage  |
|              | 9,21,33,45,57,69,81,93  | Log of fatigue damage                                 |
|              | 12,24,36,48,60,72,84,96 | Scale factor from Factor of Safety analysis           |

**Remarks:**

1. CTRIA6 and CTRIAR item codes are listed for Grids 1-3, respectively.
2. CQUAD4, CQUAD8, CQUADR, CTETRA item codes are listed for Grids 1-4, respectively.
3. CPYRAM item codes are listed for Grids 1-5, respectively.
4. CPENTA item codes are listed for Grids 1-6, respectively.
5. CHEXA item codes are listed for Grids 1-8, respectively.
6. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARM use the same item codes in [Table 7-14](#).
7. If it is desired to use a life response (or other item) for all nodes (using LOC=NODE or NODA on FTGPARM), all item codes for the particular item are required, e.g., 4, 16, 27, and 38 for CTETRA (one DRESP1 entry for each item code).
8. For shell elements with bottom (Z1) and top (Z2) layers, if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular response for a set of elements or property sets. So as an example, to specify the bottom and top response (Life in *Repeats*) for all the grids of the specified element or property set for a CQUAD4 element, use items codes: -4,-27,-50,-73
9. These item codes are used for LOC=NODE or NODA and all other element nodal values of RECOVER on the FTGPARM entry, namely, "SGAGE", "CORNER", "BILIN", and "CUBIC" also.

Table 7-16 Item Codes for Fatigue Analysis of Spot Welds

| Element Name                              | Item Code        | Real Element Data Item  |
|---|------------------|---|
| (For Top Sheet at Angle = 0.0 degrees)    | 4 (-4) (-1004)   | Fatigue life in <i>Repeats</i> of the loading sequence<br>See remarks 2. and 3. |
|   | 5 (-5) (-1005)   | Log of fatigue life in <i>Repeats</i> of the loading sequence                   |
|   | 6 (-6) (-1006)   | Fatigue life in user defined fatigue equivalent units                           |
|   | 7 (-7) (-1007)   | Log of fatigue life in user defined fatigue equivalent units                    |
|   | 8 (-8) (-1008)   | Fatigue damage  |
|   | 9 (-9) (-1009)   | Log of fatigue damage   |
|   | 13 (-13) (-1013) | Scale factor from Factor of Safety analysis                                     |
|   | 4+12*NANGLE      | Fatigue life in <i>Repeats</i> of the loading sequence                          |
| (For Weld Nugget at Angle = 0.0 degrees)  | 5+12*NANGLE      | Log of fatigue life in <i>Repeats</i> of the loading sequence                   |
|   | 6+12*NANGLE      | Fatigue life in user defined fatigue equivalent units                           |
|   | 7+12*NANGLE      | Log of fatigue life in user defined fatigue equivalent units                    |
|   | 8+12*NANGLE      | Fatigue damage  |
|   | 9+12*NANGLE      | Log of fatigue damage   |
|   | 13+12*NANGLE     | Scale factor from Factor of Safety analysis                                     |
|   | 4+24*NANGLE      | Fatigue life in <i>Repeats</i> of the loading sequence                          |
| (For Bottom Sheet at Angle = 0.0 degrees) | 5+24*NANGLE      | Log of fatigue life in <i>Repeats</i> of the loading sequence                   |
|   | 6+24*NANGLE      | Fatigue life in user defined fatigue equivalent units                           |
|   | 7+24*NANGLE      | Log of fatigue life in user defined fatigue equivalent units                    |
|   | 8+24*NANGLE      | Fatigue damage  |
|   | 9+24*NANGLE      | Log of fatigue damage   |
|   | 13+24*NANGLE     | Scale factor from Factor of Safety analysis                                     |

**Remarks:**

1. Spot weld fatigue item code is related to NANGLE, the number of calculation angles around the weld circumference as defined on the FTGPARM entry. Item codes listed above are for the first angle at zero (0) degrees. For each subsequent angle, the item code should be increased by 12. For example, item code =16 is for "Fatigue life in *Repeats* of the loading sequence" at the Top Sheet for the second angle.

2. If item codes 4-9 or 13 are negative (e.g., -4), this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for the nugget and bottom sheet as well and for all specified angles. This is a convenient method to include responses from all locations and angles by defining only a single DRESP1 entry for any particular response for a set of spot weld elements or property sets. So as an example, to specify the response (Life in *Repeats*) for all the locations and angles, use a single DRESP1 entry with item code: -4.
3. In conjunction with the above remark, if the item code is negative and multiplied by 1000 (e.g., -1004), then the nugget responses are not generated. Only the top and bottom sheet responses are internally generated.

Table 7-17 Item Codes for Fatigue Analysis of Seam Welds

| Element Name               | Item Code      | Real Element Data Item  |
|----------------------------|----------------|---|
| CQUAD4<br>CQUAD8<br>CQUADR | 4, 19, 34, 45  | Fatigue life in <i>Repeats</i> of the loading sequence        |
|                            | 5, 20, 35, 46  | Log of fatigue life in <i>Repeats</i> of the loading sequence |
|                            | 6, 21, 36, 47  | Fatigue life in user defined fatigue equivalent units         |
|                            | 7, 22, 37, 48  | Log of fatigue life in user defined fatigue equivalent units  |
|                            | 8, 23, 38, 49  | Fatigue damage  |
|                            | 9, 24, 39, 50  | Log of fatigue damage   |
|                            | 12, 27, 42, 53 | Scale factor from Factor of Safety analysis                   |
| CTRIA6<br>CTRIAR           | 4, 19, 34      | Fatigue life in <i>Repeats</i> of the loading sequence        |
|                            | 5, 20, 35      | Log of fatigue life in <i>Repeats</i> of the loading sequence |
|                            | 6, 21, 36      | Fatigue life in user defined fatigue equivalent units         |
|                            | 7, 22, 37      | Log of fatigue life in user defined fatigue equivalent units  |
|                            | 8, 23, 38      | Fatigue damage  |
|                            | 9, 24, 39      | Log of fatigue damage   |
|                            | 12, 27, 42     | Scale factor from Factor of Safety analysis                   |

**Remarks:**

1. CQUAD4, CQUAD8, CQUADR item codes are listed for Grids 1-4, respectively.
2. CTRIA6 and CTRIAR item codes are listed for Grids 1-3, respectively.
3. Note that FTGDEF has a keyword “SEAMW” that may have the field NDSIDi, which lists seam line grids. If this is used, it means that some grids of the elements will not have fatigue life associated with them and are excluded from the analysis.

## Equivalent Radiated Power (ERP) Item Codes

Table 7-18      Equivalent Radiated Power (ERP) Item Codes

| Element Name                 | Item Code | Item         |
|------------------------------|-----------|--------------|
| QUAD4, CQUADR, CTRIA3,CTRIAR | 2         | ERP value    |
| QUAD4, CQUADR, CTRIA3,CTRIAR | 3         | ERP Fraction |
| QUAD4, CQUADR, CTRIA3,CTRIAR | 4         | ERP(DB)      |

# 8

## Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions
- Degree-of-Freedom Set Bulk Data Entries

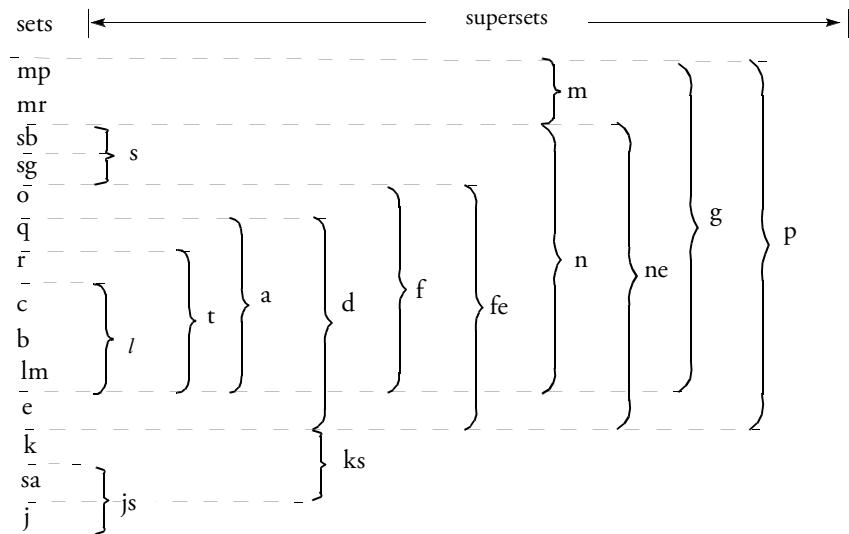
## Degree-of-Freedom Set Definitions

Each degree-of-freedom is a member of one mutually exclusive set. Set names have the following definitions:

| Set Name | Definition   |
|----------|--|
| mp       | Degrees-of-freedom eliminated by <u>multipoint</u> constraints.  |
| mr       | Degrees-of-freedom eliminated by <u>multipoint</u> constraints created by the <u>rigid</u> elements using the LGELIM method on the Case Control command RIGID.       |
| sb*      | Degrees-of-freedom eliminated by <u>single-point</u> constraints that are included in <u>boundary</u> condition changes and by the AUTOSPC feature. (See the sz set) |
| sg*      | Degrees-of-freedom eliminated by <u>single-point</u> constraints that are specified on the PS field on <u>GRID</u> Bulk Data entries.                                |
| sz       | Degrees-of-freedom eliminated by the AUTOSPC feature.  |
| o        | Degrees-of-freedom <u>omitted</u> by structural matrix partitioning.   |
| q        | Generalized degrees-of-freedom assigned to component modes and residual vectors.   |
| r        | <u>reference</u> degrees-of-freedom used to determine free body motion.  |
| c        | Degrees-of-freedom that are free during component mode synthesis or dynamic reduction.   |
| b        | Degrees-of-freedom fixed during component mode analysis or dynamic reduction.  |
| lm       | <u>Lagrange</u> <u>multiplier</u> degrees-of-freedom created by the rigid elements using the LAGR method on the Case Control command, RIGID.                         |
| e        | <u>extra</u> degrees-of-freedom introduced in dynamic analysis.  |
| sa       | Permanently constrained aerodynamic degrees-of-freedom.  |
| k        | Aerodynamic mesh point set for forces and displacements on the aero mesh.  |
| j        | Aerodynamic mesh collocation point set (exact physical interpretation is dependent on the aerodynamic theory).   |

\*Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. Since these sets are not used explicitly in the solution sequences, this need not concern the user. However, those who use these sets in their own DMAPs should avoid redundant specifications when using these sets for partitioning or merging operations. That is, a degree-of-freedom should not be specified on both a PS field of a GRID entry (sg set) and on a selected SPC entry (sb set). Redundant specifications will cause UFM 2120 in the VEC module and behavior listed in [MSC Nastran DMAP Programmer's Guide](#) for the UPARTN module. These sets are exclusive, however, from the other mutually exclusive sets.

Each degree-of-freedom is also a member of one or more combined sets called “supersets.” Supersets have the following definitions:



| Set Name         | Meaning (+ indicates union of two sets)  |
|------------------|--|
| $s = sb + sg$    | all degrees-of-freedom eliminated by single point constraints  |
| $l = b + c + lm$ | the degrees-of-freedom remaining after the reference degrees-of-freedom are removed (degrees-of-freedom left over) |
| $t = l + r$      | the total set of physical boundary degrees-of-freedom for superelements  |
| $a = t + q$      | the analysis set used in eigensolution   |
| $d = a + e$      | the set used in dynamic analysis by the direct method  |
| $f = a + o$      | unconstrained (free) structural degrees-of-freedom   |
| $fe = f + e$     | free degrees-of-freedom plus extra degrees-of-freedom  |
| $n = f + s$      | all degrees-of-freedom not constrained by multipoint constraints   |
| $ne = n + e$     | all degrees-of-freedom not constrained by multipoint constraints plus extra degrees-of-freedom                     |
| $m = mp + mr$    | all degrees-of-freedom eliminated by multipoint constraints  |
| $g = n + m$      | all degrees-of-freedom including scalar degrees-of-freedom   |
| $p = g + e$      | all physical degrees-of-freedom including extra point degree-of-freedom  |
| $ks = k + sa$    | the union of k and the re-used s-set (6 dof per grid)  |
| $js = j + sa$    | the union of j and the re-used s-set (6 dof per grid)  |

table continued on the next page

| Set Name        | Meaning (+ indicates union of two sets)   |
|-----------------|---|
| $fr = o + l$    | statically independent set minus the statically determinate supports ( $fr = f - q - r$ ) |
| $v = o + c + r$ | the set free to vibrate in dynamic reduction and component mode synthesis                 |
| $al = a - lm$   | a-set without Lagrange multiplier degree-of-freedoms.                                     |
| $dl = d - lm$   | d-set without Lagrange multiplier degree-of-freedoms.                                     |
| $gl = g - lm$   | g-set without Lagrange multiplier degree-of-freedoms.                                     |
| $ll = l - lm$   | l-set without Lagrange multiplier degree-of-freedoms.                                     |
| $nf = ne - lm$  | ne-set without Lagrange multiplier degree-of-freedoms.                                    |
| $pl = p - lm$   | p-set without Lagrange multiplier degree-of-freedoms.                                     |
| $tl = t - lm$   | t-set without Lagrange multiplier degree-of-freedoms.                                     |
| $nl = n - lm$   | n-set without Lagrange multiplier degree-of-freedoms.                                     |
| $fl = f - lm$   | f-set without Lagrange multiplier degree-of-freedoms.                                     |
| $ff = fe - lm$  | fe-set without Lagrange multiplier degree-of-freedoms.                                    |

The a-set and o-set are created in the following ways:

1. If only OMITi entries are present, then the o-set consists of degrees-of-freedom listed explicitly on OMITi entries. The remaining f-set degrees-of-freedom are placed in the b-set, which is a subset of the a-set.
2. If ASETi or QSETi entries are present, then the a-set consists of all degrees-of-freedom listed on ASETi entries and any entries listing its subsets, such as QSETi, SUPPORTi, CSETi, and BSETi entries. Any OMITi entries are redundant. The remaining f-set degrees-of-freedom are placed in the o-set.
3. If there are no ASETi, QSETi, or OMITi entries present but there are SUPPORTi entries present, then the entire f-set is placed in the a-set and the o-set is not created.
4. There must be at least one explicitly ASETi, QSETi, or OMITi entry for the o-set to exist, even if the ASETi, QSETi, or OMITi entry is redundant.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

$\xi_o$  = rigid body (zero frequency) modal degrees-of-freedom

$\xi_f$  = finite frequency modal degrees-of-freedom

$\xi_i$  =  $\xi_o + \xi_f$ , the set of all modal degrees-of-freedom

One vector set is defined that combines physical and modal degrees-of-freedom:

$u_h$  =  $\xi_i + u_e$ , the set of all modal degrees-of-freedom

The membership of each degree-of-freedom can be printed by use of the Bulk Data entries PARAM,USETPRT and PARAM,USETSEL.

## Degree-of-Freedom Set Bulk Data Entries

Degrees-of-freedom are placed in sets as specified by the user on the following Bulk Data entries:

| Name    | Bulk Data Entry Name   |
|---------|--|
| m       | MPC, MPCADD, MPCAX, POINTAX, RBAR, RBAR1, RBE1, RBE2, RBE3, RJOINT, RROD, RSPLINE, RSSCON, RTRPLT, RTRPLT1, GMBC, GMSPC* |
| sb      | SPC, SPC1, SPCADD, SPCAX, FLSYM, GMSPC*, BNDGRID, PARAM,AUTOSPC,YES  |
| sz      | PARAM,AUTOSPC,YES  |
| sg      | GRID, GRIDB, GRDSET (PS field)   |
| o       | OMIT, OMIT1, OMITAX, GRID (SEID field), SESET  |
| q       | QSET, QSET1  |
| r       | SUPPORT, SUPPORT1, SUPAX   |
| c       | CSET, CSET1, BNDFREE, BNDFRE1  |
| b       | BSET, BSET1, BNDFIX, BNDFIX1   |
| e       | EPOINT   |
| sa      | CAEROi   |
| k       | CAEROi   |
| a       | ASET, ASET1, Superelement exterior degrees-of-freedom, CSUPEXT   |
| ap      | ACCSPT   |
| rv      | RVDOF and RVDOF1   |
| u1 - u6 | USET, USET1, SEUSET, and SEUSET1   |

\*Placed in set only if constraints are not specified in the basic coordinate system.

In superelement analysis, the appropriate entry names are preceded by the letters SE, and have a field reserved for the superelement identification number. This identification is used because a boundary (exterior) grid point may be in one mutually exclusive set in one superelement and in a different set in the adjoining superelement. The SE-type entries are internally translated to the following types of entry for the referenced superelement:

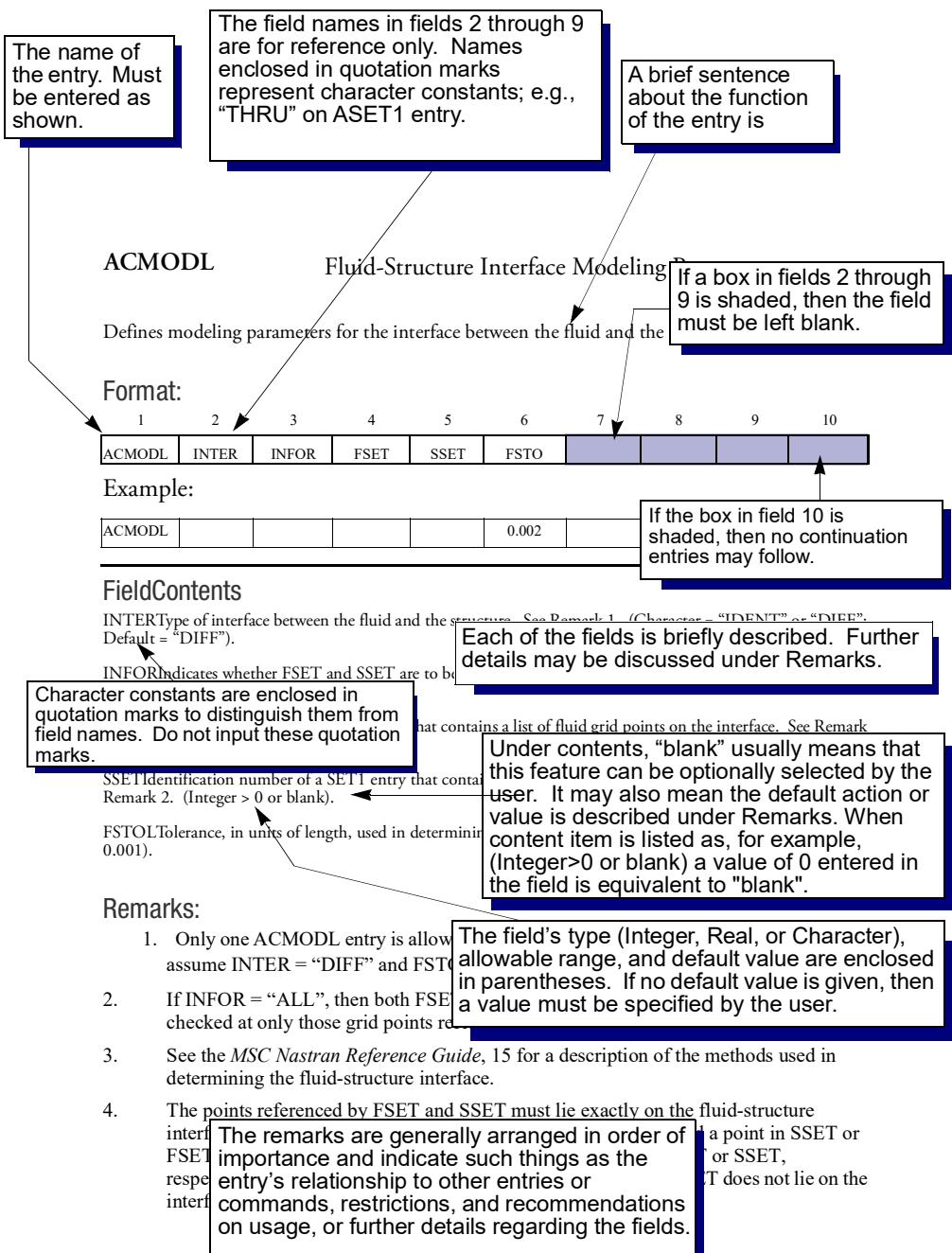
| Entry Type | Equivalent Type         |
|------------|-------------------------|
| SEQSETi    | QSETi                   |
| SESUP      | SUPPORT                 |
| SECSETi    | CSETi, BNDFREE, DNBDRE1 |
| SEBSETi    | BSETi, BNDFIXi          |

# 9

## Bulk Data Entries

- Key to Descriptions
- Bulk Data Entry Descriptions

## Key to Descriptions



## The Bulk Data Section

The Bulk Data Section contains entries that specify model geometry, element connectivity, element and material properties, constraints (boundary conditions), and loads. Some entries, such as loads and constraints, are selected by an appropriate Case Control command.

Entries are prepared in either fixed or free field format. The descriptions in this section show only the fixed format. Entries that are used by the MSGMESH program are not included in this guide. For a description of the various format options and the MSGMESH entries, see the [Use of Parameters](#) in the *MSC Nastran Reference Guide*.

New Bulk Data Entries added after the MSC Nastran 2011 version are only valid with the default option IFPSTAR=YES.

## Bulk Data Entry Descriptions

Each Bulk Data entry is described as follows:

### Description

A brief sentence about the function of the entry is given.

### Format

The name of the entry is given in the first field. The subsequent fields are described under the Field and Contents Section. Shaded fields must be left blank. If field 10 is shaded, then no continuation entries are permitted. Character strings enclosed in quotation marks must be specified without the quotation marks as shown in the example.

### Example

A typical example is given.

### Field and Contents

Each of the fields 2 through 9 that are named in the Format section is briefly described under Contents. The field's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The field must be specified by the user if no default value is given.

### Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as how the Bulk Data entry is selected in the Case Control Section, its relationship to other entries, restrictions and recommendations on its use, and further descriptions of the fields.

## Format of Bulk Data Entries

### Real, Integer, and Character Input Data

Nastran is quite particular about the input requirements for data entry. The three possible types of data entries are Integer, Real, and Character (sometimes called literal, or BCD-binary coded decimal). The three types of data are described as follows:

|           |  |
|-----------|--|
| Integer   | Cannot contain a decimal point.  |
| Real      | Must contain a decimal point. If however, system cell 444 is set to 9 and the only possible value format is real, then an integer representation (less decimal point) will be converted to real. A single decimal point is not be considered a real zero value. Either 0 or 0. (a digit) must be supplied.   |
| Character | Can be alphanumeric, and with exceptions (listed on specific bulk data entries) should always start with an alpha character. Legal alpha characters are the English: A-Z and a-z. Legal numeric characters are 0-9, and in special circumstances in fields that allow user naming the dash (-) and the underscore (_). In user naming fields use of \$, &, *, =, +, period (.), comma (,) should be avoided as they often have special meaning to the nastran bulk data interpreter. Misspelled required words such as the name of an entry will result in a fatal message. Substitution of non English alphabetic characters will cause a fatal message to be issued. |

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

|       |       |       |
|-------|-------|-------|
| 7.0   | .7E1  | 0.7+1 |
| .70+1 | 7.E+0 | 70.-1 |

### Free, Small, and Large Field Formats

Nastran has three different field formats for input data:

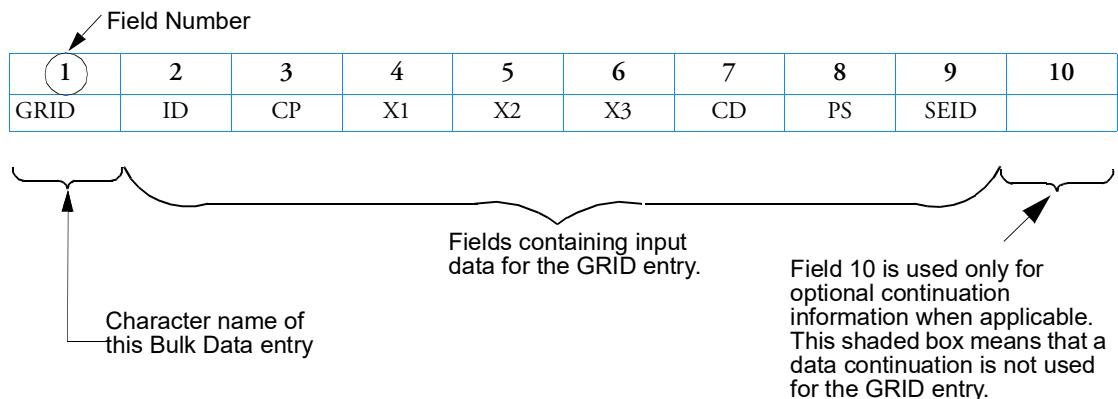
|                    |  |
|--------------------|--|
| Free Field Format  | Input data fields are separated by commas.   |
| Small Field Format | Ten fields of eight characters each.   |
| Large Field Format | Ten fields-eight fields containing actual data are sixteen characters each (fields 2-5 and 6-9). Large fields are used when greater numerical accuracy is required. See the <a href="#">Large Field Format, 1126</a> for additional information. |

The NASTRAN statement, File Management Section, Executive Control Section, and Case Control Section use free field format. The Bulk Data Section allows the use of any of the three formats.

Nastran Bulk Data contains ten fields per input data entry. The first field contains the character name of the Bulk Data item (e.g., GRID, CBAR, MAT1, etc.). Fields two through nine contain data input information

for the Bulk Data entry. The tenth field never contains data—it is reserved for entry continuation information, if applicable.

Consider the format of a typical MSC Nastran Bulk Data entry, the GRID entry, which is used in Nastran to describe the geometry of the structural model.



### Example:

| 1    | 2 | 3 | 4   | 5    | 6   | 7 | 8   | 9 | 10 |
|------|---|---|-----|------|-----|---|-----|---|----|
| GRID | 2 |   | 1.0 | -2.0 | 3.0 |   | 136 |   |    |

We will now represent this example in free field, small field, and large field formats.

### Free Field Format

In free field format, data fields are separated by commas or blanks (commas are strongly recommended). The following shows the GRID Bulk Data entry example in free field format:

GRID,2,,1.0,-2.0,3.0,,136

A bracket under the second and third commas is labeled 'Two consecutive commas indicate an empty field'.

The rules for free field format are as follows:

- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If more significant digits are needed, use the large field format. If however, NASTRAN IFPSTAR=YES is used, then the entry will automatically be converted to large field format.

- Free field data cannot contain embedded blanks to skip fields. An example of a free field embedded blank is shown:

GRID,2,,1 0,-2.0,3.0,,136



Embedded blank  
not allowed

In a continuation line, if a comma is the first character and in the first 12 columns, this line is free format.

- A dollar sign \$ can be used within a line. All data following the \$ on the line will be for comments. An example of a comment is shown:

DISP(PUNCH)=ALL \$ request punched displacement output for all grids

The free field data entry capability in Nastran have been enhanced to support easy to use data input formats. The following examples illustrate the possible forms of the free field data input and the resulting translation to the fixed-field format.

Entry with or without user continuation mnemonics.

MATT9,1101,2 ,3 ,4 ,,,8 ,+P101  
+P101,9 ,,,13

Translates to:

| 1     | 2    | 3 | 4 | 5 | 6  | 7 | 8 | 9 | 10    |
|-------|------|---|---|---|----|---|---|---|-------|
| MATT9 | 1101 | 2 | 3 | 4 |    |   |   | 8 | +P101 |
| +P101 | 9    |   |   |   | 13 |   |   |   |       |

GRID,100,,1.0,0.0,0.0,,456

Translates to:

|      |     |  |     |     |     |  |     |  |  |
|------|-----|--|-----|-----|-----|--|-----|--|--|
| GRID | 100 |  | 1.0 | 0.0 | 0.0 |  | 456 |  |  |
|------|-----|--|-----|-----|-----|--|-----|--|--|

The continuation mnemonics are not included because they are not required. This is illustrated by the entry with automatic continuation:

SPC1,100,12456,1,2,3,4,5,6,7,8,9,10

Translates to:

| 1    | 2   | 3     | 4 | 5  | 6 | 7 | 8 | 9 | 10 |
|------|-----|-------|---|----|---|---|---|---|----|
| SPC1 | 100 | 12456 | 1 | 2  | 3 | 4 | 5 | 6 |    |
|      | 7   | 8     | 9 | 10 |   |   |   |   |    |

If more than 80 characters of data are required, the free field may be continued in the next line provided that the next entry starts with a comma in the first column. The next entry will be a logical continuation of the first. For example, the free-field entry:

MATT9,1151,2 ,3 ,4 ,,,8  
,9 ,,,13

Translates to:

| 1     | 2    | 3 | 4 | 5 | 6  | 7 | 8 | 9 | 10 |
|-------|------|---|---|---|----|---|---|---|----|
| MATT9 | 1151 | 2 | 3 | 4 |    |   |   | 8 |    |
|       | 9    |   |   |   | 13 |   |   |   |    |

Which is equivalent to:

MATT9,1151,2 ,3 ,4 ,,,8 ,+  
,9 ,,,13

Translates to:

| 1     | 2    | 3 | 4 | 5 | 6  | 7 | 8 | 9 | 10 |
|-------|------|---|---|---|----|---|---|---|----|
| MATT9 | 1151 | 2 | 3 | 4 |    |   |   | 8 |    |
| +     | 9    |   |   |   | 13 |   |   |   | +  |

The free field data entry can be used to input mixed Small Field, Large Field continuations. Note that the plus (+) and asterisk (\*) characters are used to indicate Small Field and Large Field input form respectively when free field data entry is used. For example, the entries:

MATT9\*,1302,2 ,4 ,+  
+,,,,,13

Translates to:

| 1      | 2 | 3    | 4 | 5 | 6  | 7 | 8 | 9 | 10 |
|--------|---|------|---|---|----|---|---|---|----|
| MATT9* |   | 1302 |   | 2 |    |   | 4 |   | +  |
| +      |   |      |   |   | 13 |   |   |   |    |

MATT9,1303,2 ,3 ,4 ,,,8 ,+  
\*,9 ,,,,+  
\*,13

Translates to:

| 1     | 2    | 3  | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|------|----|---|---|---|---|---|---|----|
| MATT9 | 1303 | 2  | 3 | 4 |   |   |   | 8 | +  |
| *     |      | 9  |   |   |   |   |   |   |    |
| *     |      | 13 |   |   |   |   |   |   |    |

MATT9,1355,2 ,3 ,5 ,,,8 ,+  
\*,10 ,,,+  
+,17

Translates to:

| 1     | 2    | 3 | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|------|---|----|---|---|---|---|---|----|
| MATT9 | 1355 | 2 | 3  |   | 5 |   |   | 8 | +  |
| *     |      |   | 10 |   |   |   |   |   | +  |
| +     | 17   |   |    |   |   |   |   |   |    |

System cell 363 must be set to 1 (i.e., system(363)=1, or STRICTUAI=1) if more than 80 characters of data are required, then the free-field entry is continued by terminating the parent with a comma. The next entry will be a logical continuation of the first. It is not required to end the first entry at any specific point. This is illustrated by the entry:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,  
17, 18, 19, 20

Translates to:

| 1     | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|-----|-----|----|----|----|----|----|----|----|
| CHEXA | 200 | 200 | 1  | 2  | 3  | 4  | 5  | 6  |    |
|       | 7   | 8   | 9  | 10 | 11 | 12 | 13 | 14 |    |
|       | 15  | 16  | 17 | 18 | 19 | 20 |    |    |    |

Because of the feature allowing more than 10 fields of data to be entered on one free field entry, IT IS NOT ALLOWED to terminate a single free field entry with a comma. For example:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,

### Small Field Format

Small field format separates a Bulk Data entry into ten equal fields of eight characters each:

| 8 character field |    |    |    |    |    |    |    |    |    |
|-------------------|----|----|----|----|----|----|----|----|----|
| 1                 | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
| 11                | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 |
| 80 characters     |    |    |    |    |    |    |    |    |    |

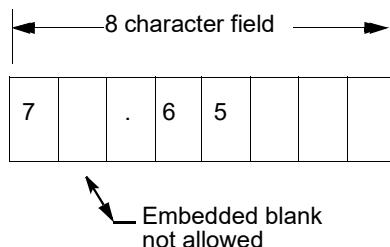
The following is an example of the GRID entry in small field format:

| 1    | 2 | 3 | 4   | 5    | 6   | 7 | 8   | 9 | 10 |
|------|---|---|-----|------|-----|---|-----|---|----|
| GRID | 2 |   | 1.0 | -2.0 | 3.0 |   | 136 |   |    |

The rules for small field format are as follows:

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right or left justified, although aligning the data fields is good practice.

- Small field input data cannot contain any embedded blanks. An example of a small field embedded blank is shown:



### Large Field Format

A high degree of numerical accuracy is required in some Nastran applications. Large field format is used when small field format does not provide enough significant digits (recall that a minus sign, decimal point, and the "E" in scientific notation count as characters).

Large field format requires (at least) two lines for each entry: the first and last field of each line contains eight columns, and the fields in between contain 16 columns. Short field becomes two lines. Large field entries are denoted by an asterisk (\*) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

The following is an example of the GRID Bulk Data entry example in large field format:

#### First Line: (Left half of single field)

| Field | 1A    | 2  | 3  | 4   | 5    | 6      |
|-------|-------|----|----|-----|------|--------|
|       | GRID* | 2  |    | 1.0 | -2.0 | GRID10 |
|       | 8     | 16 | 16 | 16  | 16   | 8      |

columns

#### Second Line: (Right half of single field)

| Field | 1B      | 6   | 7  | 8   | 9  | 10B |
|-------|---------|-----|----|-----|----|-----|
|       | *GRID10 | 3.0 |    | 136 |    |     |
|       | 8       | 16  | 16 | 16  | 16 | 8   |

columns

### Continuations

Some Bulk Data entries require more than eight fields (72 columns) of data. Continuations are required in such cases. To do this, a parent entry (the first line) is followed by one or more continuation entries on subsequent lines. For example, consider the following PBAR simple beam property entry (do not worry about what each field represents-this will be explained later):

Format:

| 1    | 2   | 3   | 4   | 5  | 6  | 7  | 8   | 9  | 10 |
|------|-----|-----|-----|----|----|----|-----|----|----|
| PBAR | PID | MID | A   | I1 | I2 | J  | NSM |    |    |
|      | C1  | C2  | D1  | D2 | E1 | E2 | F1  | F2 |    |
|      | K1  | K2  | I12 |    |    |    |     |    |    |

Continuation Example:

|      |     |     |     |      |      |     |    |    |      |
|------|-----|-----|-----|------|------|-----|----|----|------|
| PBAR | 39  | 6   | 2.9 | 1.86 | 2.92 | .48 |    |    | +PB1 |
| +PB1 | 0.  | 0.  | 0.  | 1.   | 1.   | 1.  | 1. | 0. | +PB2 |
| +PB2 | .86 | .86 |     |      |      |     |    |    |      |

+PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line. +PB2 in the second line points the third line, and so on.

Continuation fields can also be generated automatically by Nastran (this approach is the recommended practice). To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank. In the case of double-width generated continuations are not blank in field 1, but have an "\*" in column 1. Nastran will then generate unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:

```
CHEXA,      1,      10,      3,      5,      7,      1,      15,      17,
,      19,      13,      4,      6,      8,      2,      10,      11,
```

Output (.F06) file:

```
GEAR TOOTH EXAMPLESEPTEMBER 29, 1993 MSC.Nastran 9/ 4/91 PAGE 3
S O R T E D B U L K D A T A E C H O
CARD
COUNT. 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 ..
1- CHEXA      1      10      3      5      7      1      15      17      +000001
2-++000001    19      13      4      6      8      2      10      11      +000002
3-++000002    12      9      16      18      20      14      +000003
```

Blank lines are allowed if they are in between entries but not allowed if they are in an entry and further continuations are followed if ifpstar is yes, for example, the blank line in below entry is not allowed. To ignore those blank lines, set system(767) to 1.

CHEXA, 1, 10, 3, 5, 7, 1, 15, 17,

```
, 19, 13, 4, 6, 8, 2, 10, 11,
, 12, 9, 16, 18, 20, 14
```

Nastran Continuation fields (fields one and ten) are replicated using the following conventions:

- Only letters of the alphabet and integers may be used. They are coded into a base 36 number. That is, the sequence of numbers is 0,1, 2, ..., 8, 9, A, B, ...
- The first character in field one or ten is not incremented.
- The continuation fields are incremented by +1 regardless of the value specified by the user.

4. The number of characters in an incremented field will not be increased. For example, if the first field is "0", the thirty-seventh field will also be "0", resulting in an illegal entry. A method to solve this problem would be to start with a first field of "00". This will provide thirty-six squared unique fields.
5. At least one field in fields 2 through 8 of continuation entries must be non-blank.

## Replication

Replication is a limited data generation capability which may be used in a fixed or free-field format and is not currently fully supported with NASTRAN SYSTEM(444)=1 and its use, therefore, is not recommended. Also replication is not at all currently supported with SimX reading a user-generated input file and strange models may be displayed.

1. Duplication of fields from the preceding entry is accomplished by coding the symbol =.
2. Duplication of all trailing fields from the preceding entry is accomplished by coding the symbol ==.
3. Incrementing a value from the previous entry is indicated by coding \*x or \*(x), where x is the value of the increment. "x" should be a real number for real fields or an integer for integer fields.
4. Repeated replication is indicated by coding =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.
5. Data items may be enclosed with parentheses or the parentheses may be deleted.
6. The MSGMESH capability includes the capabilities described here, plus the following capabilities as long as NASTRAN MESH is specified in the File Management Section.
  - Continuation entry fields may be incremented or decremented.
  - Repeated replication is indicated by coding =(n) in field 1, where n is number of entry images to be generated using the values of increments from the current or preceding replication entry.

Entered entries:

GRID, 101, 17, 1.0, 10.5,,17,3456

= , \*1, =, \*0.2, ==

=3

Generated entries:

|      |     |    |     |      |  |    |      |  |  |
|------|-----|----|-----|------|--|----|------|--|--|
| GRID | 101 | 17 | 1.0 | 10.5 |  | 17 | 3456 |  |  |
| GRID | 102 | 17 | 1.2 | 10.5 |  | 17 | 3456 |  |  |
| GRID | 103 | 17 | 1.4 | 10.5 |  | 17 | 3456 |  |  |
| GRID | 104 | 17 | 1.6 | 10.5 |  | 17 | 3456 |  |  |
| GRID | 105 | 17 | 1.8 | 10.5 |  | 17 | 3456 |  |  |

- A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

```
BSET1,123,1,2,3,4,5,6,7
,,*7,*7,*7,*7,*7,*7
=(3)
```

Generated entries:

|         |     |    |    |    |    |    |    |    |        |
|---------|-----|----|----|----|----|----|----|----|--------|
| BSET1   | 123 | 1  | 2  | 3  | 4  | 5  | 6  | 7  | +00001 |
| ++00001 |     | 8  | 9  | 10 | 11 | 12 | 13 | 14 | +00002 |
| ++00002 |     | 15 | 16 | 17 | 18 | 19 | 20 | 21 | +00003 |
| ++00003 |     | 22 | 23 | 24 | 25 | 26 | 27 | 28 | +00004 |
| ++00004 |     | 29 | 30 | 31 | 32 | 33 | 34 | 35 | +00005 |

- A “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of 9 entries may be replicated as a group. The default continuation entry increment is 10. Example:

Entered entries:

```
CTRIA3,10,1,1,10,11/+C1
=(D),*(1),=,*(1),*(1)/*(20)
+C1,,,2.0,1.0,1.0
=(2),==
```

Generated entries:

|        |    |   |     |     |     |  |  |  |      |
|--------|----|---|-----|-----|-----|--|--|--|------|
| CTRIA3 | 10 | 1 | 1   | 10  | 11  |  |  |  | +C1  |
| +C1    |    |   | 2.0 | 1.0 | 1.0 |  |  |  |      |
| CTRIA3 | 11 | 1 | 1   | 11  | 12  |  |  |  | +C21 |
| +C21   |    |   | 2.0 | 1.0 | 1.0 |  |  |  |      |
| CTRIA3 | 12 | 1 | 1   | 12  | 13  |  |  |  | +C41 |
| +C41   |    |   | 2.0 | 1.0 | 1.0 |  |  |  |      |

- Parentheses are optional on replication entries and an equal sign may replace an asterisk.

The following is an example of the use of replication, automatic continuation field generation, and the free field format:

GRID, 101, 17, 1.0, 10.5,,17,3456

=,\*1,=,\*0.2, \*(0.1), == \$ COMMENTS MAY APPEAR AFTER \$

=3

EIGR,13,GIV,,30.

,MASS

CBAR,1 ,1 ,101 ,102,0.,0.,1.,,+0

=,\*1,=,\*1,\*1====\*1

+0,56

\*1,=\$

The above free-field entries will generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

Note: A “,” should always be used after the “\*1” for the continuation increment even if fixed field format is being used.

|          |      |     |     |      |    |    |      |  |          |
|----------|------|-----|-----|------|----|----|------|--|----------|
| CBAR     | 1    | 1   | 101 | 102  | 0. | 0. | 1.   |  | +0       |
| +0       | 56   |     |     |      |    |    |      |  |          |
| CBAR     | 2    | 1   | 102 | 103  | 0. | 0. | 1.   |  | +1       |
| +1       | 56   |     |     |      |    |    |      |  |          |
| EIGR     | 13   | GIV |     | 30.  |    |    |      |  | +0000001 |
| ++000001 | MASS |     |     |      |    |    |      |  |          |
| GRID     | 101  | 17  | 1.0 | 10.5 |    | 17 | 3456 |  |          |
| GRID     | 102  | 17  | 1.2 | 10.6 |    | 17 | 3456 |  |          |
| GRID     | 103  | 17  | 1.4 | 10.7 |    | 17 | 3456 |  |          |
| GRID     | 104  | 17  | 1.6 | 10.8 |    | 17 | 3456 |  |          |
| GRID     | 105  | 17  | 1.8 | 10.9 |    | 17 | 3456 |  |          |

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user's responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

## Identifiers

MSC Nastran is designed to use Identifiers for many quantities, the most common include:

|     |                                |
|-----|--------------------------------|
| ID  | Grid Identification number     |
| EID | Element Identification number  |
| PID | Property Identification number |
| MID | Material Identification number |
| CID | Coordinate System number       |

When these identifications are given they must be unique in their class. All IDs defined on GRID, GRIDB, GRIDF, GRIDS, SPOINT and EPOINT must be unique.

In the case of Element Identification (EID), elements are generally given by "C" entries, such as CBAR, CBEAM, CQUAD, CHEXA, but rigid elements such as RBAR, RBE1, RBE2 are also treated as elements with respect to the EID, and should also be part of this class. All EID must be unique.

All Identification numbers must be greater than zero ( example EID > 0)

Furthermore Element Identification and Grid Identification must be less than 100,000,000 to insure data integrity.

There are two types of Property entries, primary ones and secondary ones. The property id on the primary property entries must be unique. These property identifier numbers are referenced on either "C" options defining element connectivity or "G" options for p-element technology. The correspondence between these options is given below.

| Referenced by | Primary Property Names                                      |
|---------------|---|
| CHBDYP        | BDYOR   |
| CHBDYG        | BDYOR   |
| CHBDYE        | BDYOR   |
| CAABSF        | PAABSF  |
| CACINF3       | PACINF  |
| CACINF4       | PACINF  |
| CAERO1        | PAERO1  |
| CAERO2        | PAERO2  |
| CAERO3        | PAERO3  |
| CAERO4        | PAERO4  |
| CAERO5        | PAERO5  |
| CAXISYM       | PAXISYM   |
| CBAR          | PBAR,PBRL, PBRSECT  |
| CBEAM         | PBEAM, PBCOMP, PBEAML, PBMSECT, PDISCR,<br>PBMARB6, PMMNUM6 |
| CBEAM3        | PBEAM3, PBMSECT   |
| CBELT         | PBELTD  |
| CBEND         | PBEND   |
| CBUSH         | PBUSH, PBUSHT   |
| CBUSH1D       | PBUSH1D   |
| CBUSH2D       | PBUSH2D   |
| CCONEAX       | PCONEAX   |
| CDAMP1        | PDAMP   |
| CDAMP1D       | PDAMP   |
| CDAMP3        | PDAMP   |
| CDAMP5        | PDAMP5  |
| CDUMi         | PDUMi   |
| CELAS1        | PELAS1  |
| CELAS1D       | PELAS1  |
| CELAS3        | PELAS   |
| CFAST         | PFAST   |
| CGAP          | PGAP  |

| Referenced by | Primary Property Names                              |
|---------------|---|
| CHACAB        | PACABS  |
| CHACBR        | PACBAR  |
| CHBDYP        | PHBDY   |
| CHEXA         | PSOLID, PLSOLID, PCOMPLS                            |
| CIFHEX        | PCOHE   |
| CIFPENT       | PCOHE   |
| CIFQDT        | PCOHE   |
| CIFQUAD       | PCOHE   |
| CMARKB2       | PMARKER   |
| CMARKN1       | PMARKER   |
| CMASS1        | PMASS   |
| CMASS3        | PMASS   |
| CMREBAI       | PMREBAI   |
| CMREBAR       | PMREBAR   |
| CONSPOT       | PBSPOT  |
| CPENTA        | PSOLID, PLSOLID                                     |
| CQUAD         | PLPLANE,PLCOMP                                      |
| CQUAD4        | PSHELL,PCOMP,PCOMPG, PLPLANE,PSHELL1,PSHELLD,PLCOMP |
| CQUAD8        | PSHELL,PCOMP,PCOMPG,PLPLANE,PLCOMP                  |
| CQUADR        | PSHELL,PCOMP,PCOMPG,PLPLANE                         |
| CQUADX        | PAXSYMH, PLPLANE,PLCOMP                             |
| CRAC2D        | PRAC2D  |
| CRAC3D        | PRAC3D  |
| CROD          | PROD,PBELTD   |
| CSEAM         | PSEAM   |
| CSHEAR        | PSHEAR  |
| CSPH          | PSPH  |
| CSPR          | PSPRMAT   |
| CSSHLL        | PSSHL   |
| CSSHLLH       | PSSHL   |
| CSSHLP        | PSSHL   |
| CTETRA        | PSOLID, PLSOLID                                     |
| CTQUAD        | PTSHELL   |

| Referenced by | Primary Property Names                |
|---------------|---------------------------------------|
| CTRIA3        | PSHELL,PCOMP,PCOMPG, PLPLANE, PSHELL1 |
| CTRIA6        | PSHELL,PCOMP,PCOMPG,PLPLANE           |
| CTRIAR        | PSHELL,PCOMP,PCOMPG, PLPLANE          |
| CTRIAX        | PAXSYMH                               |
| CTTRIA        | PTSHELL                               |
| CTUBE         | PTUBE                                 |
| CVISC         | PVISC                                 |
| CWELD         | PWELD                                 |
| GMINTC        | PINTC                                 |
| GMINTS        | PINTS                                 |

The secondary Property entries use the same property id as the primary property id. The relationship between these two is shown below.

| Primary | Secondary |
|---------|-----------|
| PSOLID  | PSLDN1    |
| PLPLANE | PSHLN2    |
| PCOMP   | PCOMPA    |
| PBAR    | PBARN1    |
| PBEAM   | PBEAM71   |
| PBEAM   | PBEMN1    |
| PBEAML  | PBEMN1    |
| PCOMP   | PCOMPF    |
| PCOMPG  | PCOMPF    |
| PDAMP   | PDAMPT    |
| PELAS   | PELAST    |
| PROD    | PROND1    |
| PSHEAR  | PSHEARN   |
| PSHELL  | PSHLN1    |
| PCOMP   | PSHLN1    |

There are two types of Material entries, primary ones and secondary ones. The material id on the primary material entries must be unique. The material id on the secondary Material entry must agree with the material

id on the associated primary Material entry. The correlation between primary and secondary Material entries is given below.

| Material Model            | Primary | Secondary |         |       |       |        |      |       |       |        |       |        |  |
|---------------------------|---------|-----------|---------|-------|-------|--------|------|-------|-------|--------|-------|--------|--|
| Isotropic                 | MAT1    | MATT1     | MATS1   | MATVP | MATEP | MATTEP | MATF | MATTF | MATF1 | MATFTG | MATVE | MATTVE |  |
| Shell Anisotropic         | MAT2    | MATT2     |         | MATVP | MATEP | MATTEP | MATF | MATTF |       |        |       |        |  |
| Planar Orthotropic        | MAT3    | MATT3     | MATS3   | MATVP | MATEP | MATTEP | MATF | MATTF |       |        |       |        |  |
| Hypoelastic User Sub.     | MATUSR  | MATTUSR   |         |       |       |        |      |       |       |        |       |        |  |
| General Orthotropic       | MATORT  | MATTORT   | MATSORT |       |       |        | MATF | MATTF |       |        |       |        |  |
| Shell Orthotropic         | MAT8    | MATT8     | MATS8   |       |       |        | MATF | MATTF |       |        |       |        |  |
| General Anisotropic       | MAT9    | MATT9     |         |       |       |        |      |       |       |        |       |        |  |
| Genel Hyperelastic        | MATHE   | MATTHE    |         |       |       |        |      |       |       |        | MATVE | MATTVE |  |
| Gasket                    | MATG    | MATTG     |         |       |       |        |      |       |       |        |       |        |  |
| 5th order Mooney-Rivlin   | MATHP   |           |         |       |       |        |      |       |       |        |       |        |  |
| Advanced NLELAST          | MATNLE  |           |         |       |       |        |      |       |       |        |       |        |  |
| Shape Memory              | MATSMA  |           |         |       |       |        |      |       |       |        |       |        |  |
| Isotropic Poroelastic     | MATPE1  |           |         |       |       |        |      |       |       |        |       |        |  |
| CZM Material SOL 400      | MCOHE   |           |         |       |       |        |      |       |       |        |       |        |  |
| CZM Material SOL 600      | MDELAM  |           |         |       |       |        |      |       |       |        |       |        |  |
| Composite Mixture         | MIXTURE |           |         |       |       |        |      |       |       |        |       |        |  |
| NLELAST                   | MATNLE  |           |         |       |       |        |      |       |       |        |       |        |  |
| Digimat Composite         | MATDIGI |           |         |       |       |        |      |       |       |        |       |        |  |
| Isotropic Heat Transfer   | MAT4    | MATT4     |         |       |       |        |      |       |       |        |       |        |  |
| Anisotropic Heat Transfer | MAT5    | MATT5     |         |       |       |        |      |       |       |        |       |        |  |

## Bulk Data Entry Summary

This section contains a summary of all Bulk Data entries. The entries are categorized as Geometry, Elements, Material Properties, Constraints, Loads, Solution Control, and Miscellaneous. Entries that are exclusive to SOL 600 have been grouped together at the end of the summary. Entries that are exclusive to Explicit Nonlinear analysis (SOL 700) have also been grouped together at the end of the summary.

# Constraints and Partitioning

## Component Mode Boundary Conditions

|                |  |
|----------------|--|
| <b>BNDFIX</b>  | Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations. |
| <b>BNDFIX1</b> | Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations. |
| <b>BNDFREE</b> | Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.           |
| <b>BNDFRE1</b> | Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.           |
| <b>BSET</b>    | Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.                                  |
| <b>BSET1</b>   | Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.                                  |
| <b>CSET</b>    | Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.  |
| <b>CSET1</b>   | Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.  |
| <b>QSET</b>    | Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.  |
| <b>QSET1</b>   | Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.  |
| <b>SEBSET</b>  | Defines boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.  |
| <b>SEBSET1</b> | Defines fixed boundary points for superelement.  |
| <b>SECSET</b>  | Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.   |
| <b>SECSET1</b> | Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.   |
| <b>SENQSET</b> | Defines number of internally generated scalar points for superelement dynamic reduction.   |
| <b>SEQSET</b>  | Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.   |
| <b>SEQSET1</b> | Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.   |
| <b>SESUP</b>   | Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.  |

## Free Body Supports

|          |  |
|----------|--|
| CYSUP    | Defines fictitious supports for cyclic symmetry analysis.                                  |
| SUPAX    | Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis. |
| SUPPORTi | Defines degrees-of-freedom for determinate reactions.                                      |

## Multipoint Constraints

|         |   |
|---------|---|
| MONSUM  | Defines a new monitor result that is the weighted sum of existing monitor results.                    |
| MPC     | Defines a linear relationship for two or more degrees-of-freedom.                                     |
| MPCADD  | Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries. |
| MPCD    | Defines a load selectable value for nonhomogeneous multi-point constraint.                            |
| MPCY    | Defines a linear nonhomogeneous relationship for two or more degrees-of-freedom.                      |
| MPCAX   | Defines multipoint constraints for conical shell problems.  |
| POINTAX | Defines multipoint constraints for point on conical shell.  |
| RBAR    | Defines multipoint constraints for rigid bar.   |
| RBEi    | Defines multipoint constraints for RBE1, RBE2, RBE3.  |
| RROD    | Defines multipoint constraints for rigid rod.   |
| RSPLINE | Defines multipoint constraints for spline element.  |
| RTRPLT  | Defines multipoint constraints for rigid triangular plate.  |

## p-element Geometry Constraints

|       |  |
|-------|--|
| GMBC  | Defines enforced displacements for GRID, FEEDGE, FEFACE, GMCURV, and GMSURF entries. |
| GMSPC | Defines constraints for entries.   |

## Partitioning

|         |  |
|---------|--|
| ASET    | Defines degrees-of-freedom in the analysis set (a-set).  |
| ASET1   | Defines degrees-of-freedom in the analysis set (a-set).  |
| CSUPEXT | Assigns exterior points to a superelement.   |
| GRID    | Defines interior points for a superelement.  |
| OMIT    | Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).                             |
| OMIT1   | Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).                             |
| OMITAX  | Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).                             |
| RELEASE | Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement. |

|              |   |
|--------------|---|
| <b>SEELT</b> | Reassigns superelement boundary elements to an upstream superelement. |
| <b>SESET</b> | Defines interior grid points for a superelement.                      |

### Single Point Constraints

|                |   |
|----------------|---|
| <b>FLSYM</b>   | Symmetry control for boundary in axisymmetric fluid problem.  |
| <b>GRID</b>    | Includes single point constraint definition.  |
| <b>GRIDB</b>   | Includes single point constraint definition.  |
| <b>GRDSET</b>  | Includes default for single point constraints.  |
| <b>SPC</b>     | Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis). |
| <b>SPC1</b>    | Defines a set of single point constraints.  |
| <b>SPCADD</b>  | Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.  |
| <b>SPCAX</b>   | Defines a set of single-point constraints or enforced displacements for conical shell coordinates.  |
| <b>SPCOFF</b>  | Defines degrees-of-freedom to be excluded from the AUTOSPC operation.   |
| <b>SPCOFF1</b> | Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation.  |

### User Sets

|                |   |
|----------------|---|
| <b>DEFUSER</b> | Defines new names for degree-of-freedom sets.       |
| <b>SEUSER</b>  | Defines a degree-of-freedom set for a superelement. |
| <b>SEUSER1</b> | Defines a degree-of-freedom set for a superelement. |
| <b>USER</b>    | Defines a degree-of-freedom set.                    |
| <b>USER1</b>   | Defines a degrees-of-freedom set.                   |

## Elements

A summary of the capabilities and characteristics of the small strain elements is available in [Element Summary -- Conventional MSC Nastran Elements](#) in the *MSC Nastran Reference Guide*.

### Aerodynamic Elements

|                |   |
|----------------|---|
| <b>AEFACT</b>  | Defines real numbers for aeroelastic analysis.  |
| <b>AELINK</b>  | Defines relationships between or among AESTAT and AESURF entries.   |
| <b>AELIST</b>  | Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity. Also defines server specific integer data for external spline methods. |
| <b>AEQUAD4</b> | Defines the connectivity of a quadrilateral aerodynamic element.  |

|                         |   |
|-------------------------|---|
| <a href="#">AESTAT</a>  | Specifies rigid body motions to be used as trim variables in static aeroelasticity.   |
| <a href="#">AESURF</a>  | Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.  |
| <a href="#">AESURFS</a> | Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry.             |
| <a href="#">AETRIA3</a> | Defines the connectivity of a triangular aerodynamic element.   |
| <a href="#">CAERO1</a>  | Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords.  |
| <a href="#">CAERO2</a>  | Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.  |
| <a href="#">CAERO3</a>  | Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface. |
| <a href="#">CAERO4</a>  | Defines an aerodynamic macro element for Strip theory.  |
| <a href="#">CAERO5</a>  | Defines an aerodynamic macro element for Piston theory.   |
| <a href="#">CSSCHD</a>  | Defines a scheduled control surface deflection as a function of Mach number and angle of attack.  |
| <a href="#">PAERO1</a>  | Defines associated bodies for the panels in the Doublet-Lattice method.   |
| <a href="#">PAERO2</a>  | Defines the cross-sectional properties of aerodynamic bodies.   |
| <a href="#">PAERO3</a>  | Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.               |
| <a href="#">PAERO4</a>  | Defines properties of each strip element for Strip theory.  |
| <a href="#">PAEROS</a>  | Defines properties of each strip element for Piston theory.   |

**Aerodynamic to Structure Interconnection**

|                         |   |
|-------------------------|---|
| <a href="#">AELISTC</a> | Defines a list of 8-character strings.  |
| <a href="#">SET1</a>    | Defines a list of structural grid points.   |
| <a href="#">SET2</a>    | Defines a list of structural grid points in terms of aerodynamic macro elements.  |
| <a href="#">SET3</a>    | Defines a list of grids, elements or points.  |
| <a href="#">SPBLND1</a> | Defines a strip based blending of two splines.  |
| <a href="#">SPBLND2</a> | Defines a curve based blending of two splines.  |
| <a href="#">SPLINE1</a> | Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points. |
| <a href="#">SPLINE2</a> | Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.    |
| <a href="#">SPLINE3</a> | Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.   |

|                |  |
|----------------|--|
| <b>SPLINE4</b> | Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method. |
| <b>SPLINE5</b> | Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points.  |
| <b>SPLINE6</b> | Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.  |
| <b>SPLINE7</b> | Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.   |
| <b>SPLEXIN</b> | Defines the input for a spline that will be evaluated with a user-supplied procedure.  |
| <b>SPRINRB</b> | Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.   |
| <b>SPRELAX</b> | Defines relaxation of a spline based on an adjacent spline.  |

### Axisymmetric Elements

|                |  |
|----------------|--|
| <b>CAXISYM</b> | Defines an axisymmetric shell.   |
| <b>CCONEAX</b> | Defines a conical shell element.   |
| <b>CQUADX</b>  | Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis. |
| <b>CTRIAX</b>  | Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.       |
| <b>CTRIAX6</b> | Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.  |
| <b>MREVERS</b> | Defines which elements, if any, required node numbering to be reversed in SOL 600.   |
| <b>PCONEAX</b> | Defines the properties of a conical shell element described on a CCONEAX entry.  |

### Cohesive Zone Modeling Elements

|                |   |
|----------------|---|
| <b>CIFHEX</b>  | Defines a solid interface cohesive zone modeling element in SOL 400.                |
| <b>CIFPENT</b> | Defines a solid interface cohesive zone modeling element in SOL 400.                |
| <b>CIFQDX</b>  | Defines an axisymmetric interface cohesive zone modeling element in SOL 400.        |
| <b>CIFQUAD</b> | Defines a planar interface element for cohesive zone modeling element in SOL 400.   |
| <b>MCOHE</b>   | Defines damage and cohesive energy for interface zone modeling elements in SOL 400. |
| <b>PCOHE</b>   | Defines the properties of a cohesive interface zone modeling elements in SOL 400.   |

### Connector Elements

|              |  |
|--------------|--|
| <b>CFAST</b> | Defines a fastener with material orientation connecting two surface patches. |
| <b>CSEAM</b> | Defines a seam-line connecting two surfaces.                                 |

|                         |  |
|-------------------------|--|
| <a href="#">CSLOT3</a>  | Defines an element connecting three points that solve the wave equation in two dimensions. |
| <a href="#">CSLOT4</a>  | Defines an element connecting four points that solve the wave equation in two dimensions.  |
| <a href="#">CWELD</a>   | Defines a weld or fastener connecting two surface patches or points.                       |
| <a href="#">PFAST</a>   | Defines the CFAST fastener property values.  |
| <a href="#">PSEAM</a>   | Defines the CSEAM fastener property values.  |
| <a href="#">PWELD</a>   | Defines the property of connector (CWELD) elements.  |
| <a href="#">SWLDPRM</a> | Overrides default values of parameters for connector search.                               |

### **Slidelines Contact (SOL 106, 129)**

|                        |   |
|------------------------|---|
| <a href="#">BCOMP</a>  | Defines the parameters for a contact region and its properties.   |
| <a href="#">BFRIC</a>  | Defines frictional properties between two bodies in contact.  |
| <a href="#">BLSEG</a>  | Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.             |
| <a href="#">BWIDTH</a> | Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry. |

### **Gap Elements**

|                      |   |
|----------------------|---|
| <a href="#">CGAP</a> | Defines a gap or friction element.                      |
| <a href="#">PGAP</a> | Defines the properties of the gap element (CGAP entry). |

### **Crack Tip Elements**

|                        |  |
|------------------------|--|
| <a href="#">CRAC2D</a> | Defines a two-dimensional crack tip element.   |
| <a href="#">CRAC3D</a> | Defines a three-dimensional crack tip element.   |
| <a href="#">PRAC2D</a> | Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element. |
| <a href="#">PRAC3D</a> | Defines the properties of the CRAC3D structural element.   |

### **Damping Elements**

|                         |  |
|-------------------------|--|
| <a href="#">CBUSH1D</a> | See line elements.   |
| <a href="#">CDAMP1</a>  | Defines a scalar damper element.   |
| <a href="#">CVISC</a>   | Defines a viscous damper element.  |
| <a href="#">DAMPING</a> | Specifies the values for parameter damping and/or selects optional HYBRID damping. |
| <a href="#">HYBDAMP</a> | Specifies hybrid damping parameters.   |
| <a href="#">PBUSH1D</a> | See line elements.   |

|                         |  |
|-------------------------|--|
| <a href="#">PDAMP</a>   | Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.   |
| <a href="#">PDAMP5</a>  | Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only. |
| <a href="#">PDAMPT</a>  | Defines the frequency-dependent properties for a PDAMP Bulk Data entry.  |
| <a href="#">PVISC</a>   | Defines properties of a one-dimensional viscous damping element (CVISC entry).   |
| <a href="#">ROTHYBD</a> | Defines Hybrid damping for rotordynamics.  |

## Dummy Elements

|                       |   |
|-----------------------|---|
| <a href="#">ADUMi</a> | Defines attributes of the dummy elements ( $1 \leq i \leq 9$ )                                  |
| <a href="#">CDUMi</a> | Defines a dummy element ( $1 \leq i \leq 9$ )   |
| <a href="#">PDUMi</a> | Defines the properties of a dummy element ( $1 \leq i \leq 9$ ). Referenced by the CDUMi entry. |
| <a href="#">POTEL</a> | Defines a one-dimensional dummy element for use in plotting.                                    |

## Fluid and Acoustic Elements

|                         |   |
|-------------------------|---|
| <a href="#">CAABSF</a>  | Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis. |
| <a href="#">CACINFi</a> | Defines few types of acoustic infinite elements.  |
| <a href="#">CAXIFI</a>  | Defines an axisymmetric fluid element that connects $i = 2, 3$ , or 4 fluid points.           |
| <a href="#">CAXISYM</a> | Defines two or three node axisymmetric thick shell elements.                                  |
| <a href="#">CFLUIDi</a> | Defines three types of fluid elements for an axisymmetric fluid model.                        |
| <a href="#">CHACAB</a>  | Defines the acoustic absorber element in coupled fluid-structural analysis.                   |
| <a href="#">CHACBR</a>  | Defines the acoustic barrier element.   |
| <a href="#">CHEXA</a>   | Connection definition for a pentahedron element in coupled fluid-structural analysis.         |
| <a href="#">CPENTA</a>  | Connection definition for a tetrahedron element in coupled fluid-structural analysis.         |
| <a href="#">CSLOTi</a>  | Defines slot element for acoustic cavity analysis.  |
| <a href="#">CTETRA</a>  | Defines the connections of the four-sided solid element with four to ten grid points.         |
| <a href="#">ELIST</a>   | Defines a list of structural elements for virtual fluid mass.                                 |
| <a href="#">MAT10</a>   | Fluid Material Property Definition.   |
| <a href="#">PAABSF</a>  | Defines the properties of a frequency-dependent acoustic absorber element.                    |
| <a href="#">PACABS</a>  | Defines the properties of the acoustic absorber element.                                      |
| <a href="#">PACBAR</a>  | Defines the properties of the acoustic barrier element.                                       |
| <a href="#">PACINF</a>  | Defines the properties of acoustic infinite elements.   |
| <a href="#">PANEL</a>   | Selects the set of structural grid points that define one or more panels.                     |

|                        |  |
|------------------------|--|
| <a href="#">PSOLID</a> | Defines the fluid properties of solid elements (CHEXA, CPENTA, and CTETRA entries).                                |
| <a href="#">SET1</a>   | Defines a list of structural grid points for aerodynamic analysis, XY-plots for SORT1 output, and the PANEL entry. |

### **Equivalent Radiated Power**

|                        |                                      |
|------------------------|--------------------------------------|
| <a href="#">ERPPNL</a> | Equivalent Radiated Power Definition |
|------------------------|--------------------------------------|

### **Poroelasticity (PEM)**

|                        |  |
|------------------------|--|
| <a href="#">ACTRIM</a> | ACTRAN Trimmed Material Matrices for SOL 108/111       |
| <a href="#">ACLOAD</a> | ACTRAN Acoustic Pressure Load Matrices for SOL 108/111 |
| <a href="#">MATPE1</a> | Isotropic Poroelastic Material Property definition     |
| <a href="#">MATF1</a>  | Frequency Dependent Isotropic Material Definition      |

### **Heat Transfer Elements**

|                         |  |
|-------------------------|--|
| <a href="#">BDYOR</a>   | Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.   |
| <a href="#">CHBDYi</a>  | Connection definition for surface element (CHBDYE, CHBDYG, CHBDYP).  |
| <a href="#">CONTRLT</a> | Thermal control element for heat transfer analysis.  |
| <a href="#">PHBDY</a>   | A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements. |

The PRODN1, PSHLN1, PSHLN2, PSLDN1, PSHEARN, PLCOMP, and PCOMPLS may be used to extend the nonlinear capabilities of heat transfer elements in SOL 400. PCOMPLS may also be used to extend the nonlinear capabilities of heat transfer elements in all linear solutions between SOL101 and SOL112.

The following elastic elements may also be used as heat conduction elements:

|               |  |
|---------------|--|
| Linear:       | CBAR, CROD, CONROD, CTUBE, CBEAM, CBEND. |
| Membrane:     | CTRIA3, CTRIA6, CQUAD4, CQUAD8.          |
| Axisymmetric: | CTRIAX6.                                 |
| Solid:        | CTETRA, CHEXA, CPENTA.                   |

### **Line Elements**

|                        |   |
|------------------------|---|
| <a href="#">BAROR</a>  | Default for orientation and property for CBAR.                                |
| <a href="#">BEAMOR</a> | Defines default values for field 3 and fields 6 through 8 of the CBEAM entry. |
| <a href="#">CBAR</a>   | Defines a simple beam element.  |
| <a href="#">CBEAM</a>  | Defines a beam element.   |

|                         |   |
|-------------------------|---|
| <a href="#">CBEAM3</a>  | Defines a three-node beam element.  |
| <a href="#">CBEND</a>   | Defines a curved beam, curved pipe, or elbow element.   |
| <a href="#">CBUSH1D</a> | Defines the connectivity of a one-dimensional spring and viscous damper element.                                      |
| <a href="#">CFAST</a>   | Defines a fastener with material orientation connecting two surface patches.  |
| <a href="#">CINTC</a>   | Defines a line interface element with specified boundaries.   |
| <a href="#">CMREBAI</a> | Defines Rebar elements and matching “Matrix” solid element using the Marc REBAR with INSERT Method. (SOL 600)         |
| <a href="#">CMREBAR</a> | Defines Rebar elements with matching “Matrix” solid elements using the Marc REBAR without INSERT Method. (SOL 600)    |
| <a href="#">CONROD</a>  | Defines a rod element without reference to a property entry.  |
| <a href="#">CROD</a>    | Defines a tension-compression-torsion element.  |
| <a href="#">CTUBE</a>   | Defines a tension-compression-torsion tube element.   |
| <a href="#">CWELD</a>   | Defines a weld or fastener connecting two surface patches or points.  |
| <a href="#">PBAR</a>    | Defines the properties of a simple beam element (CBAR entry).   |
| <a href="#">PBARL</a>   | Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.                           |
| <a href="#">PBARN1</a>  | Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry.                           |
| <a href="#">PBCOMP</a>  | Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. |
| <a href="#">PBEAM</a>   | Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.              |
| <a href="#">PBEAM3</a>  | Defines the properties of a three-node beam element (CBEAM3 entry).   |
| <a href="#">PBEAML</a>  | Defines the properties of a beam element by cross-sectional dimensions.   |
| <a href="#">PBEMN1</a>  | Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry.                         |
| <a href="#">PBEND</a>   | Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).                                 |
| <a href="#">PFAST</a>   | Defines the CFAST fastener property values.   |
| <a href="#">PBMSECT</a> | Defines the shape of arbitrary cross-section for CBEAM element.   |
| <a href="#">PBRSECT</a> | Defines the shape of arbitrary cross-section for CBAR element.  |
| <a href="#">PBUSH1D</a> | Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).               |
| <a href="#">PBUSH2D</a> | Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).                                 |
| <a href="#">PMREBAI</a> | Defines Rebar property information for CMREBAI elements. (SOL 600)  |
| <a href="#">PMREBAR</a> | Defines Rebar property information for CMREBAR elements. (SOL 600)  |

|                        |   |
|------------------------|---|
| <a href="#">PROD</a>   | Defines the properties of a rod element (CROD entry).                           |
| <a href="#">PRODN1</a> | Defines nonlinear property extensions for the PROD in SOL 400.                  |
| <a href="#">PTUBE</a>  | Defines the properties of a thin-walled cylindrical tube element (CTUBE entry). |
| <a href="#">PWELD</a>  | Defines the properties of connector (CWELD) elements.                           |

## Mass Elements

|                        |   |
|------------------------|---|
| <a href="#">CMASSi</a> | Connection definition for scalar mass, also property definition for i=2 or 4. |
| <a href="#">CONM1</a>  | Defines a 6 x 6 symmetric mass matrix at a geometric grid point.              |
| <a href="#">CONM2</a>  | Defines concentrated mass at a grid point.                                    |
| <a href="#">PMASS</a>  | Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries). |
| <a href="#">NSM</a>    | Non Structural Mass by ID.  |
| <a href="#">NSM1</a>   | Non Structural Mass (alternate form).   |
| <a href="#">NSMADD</a> | Non Structural Mass Set Combination.  |
| <a href="#">NSML</a>   | Lumped Non Structural Mass by ID.   |
| <a href="#">NSML1</a>  | Lumped Non Structural Mass (alternate form).                                  |

## p-element Interface Elements

|                        |  |
|------------------------|--|
| <a href="#">GMINTC</a> | Defines a p-interface element along a curve.   |
| <a href="#">GMINTS</a> | Defines a p-interface element along a surface. |
| <a href="#">PINTC</a>  | Property definition for GMINTC.                |
| <a href="#">PINTS</a>  | Property definition for GMINTS.                |

## Rigid Elements

|                         |  |
|-------------------------|--|
| <a href="#">RBAR</a>    | Defines a rigid bar with six degrees-of-freedom at each end.   |
| <a href="#">RBAR1</a>   | Alternative format for RBAR.   |
| <a href="#">RBE1</a>    | Defines a rigid body connected to an arbitrary number of grid points.  |
| <a href="#">RBE2</a>    | Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points. |
| <a href="#">RBE3</a>    | Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.   |
| <a href="#">RJOINT</a>  | Defines a rigid joint element connecting two coinciding grid points.   |
| <a href="#">RROD</a>    | Defines a pin-ended element that is rigid in translation.  |
| <a href="#">RSPLINE</a> | Defines multipoint constraints for the interpolation of displacements at grid points.  |
| <a href="#">RSSCON</a>  | Defines multipoint constraints to model clamped connections of shell-to-solid elements.  |

- RTRPLT** Defines a rigid triangular plate.  
**RTRPLT1** Defines a rigid triangular plate (alternate).

### Scalar and Bushing Elements

- CBUSH** Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.
- CBUSH1D** Defines the connectivity of a one-dimensional spring and viscous damper element.
- CBUSH2D** Defines the connectivity of a two-dimensional Linear-Nonlinear element.
- CELASi** Connection definition for scalar spring, also property definition for i=2 or 4.
- GENEL** Defines a general element.
- MGRSPR** Defines grids to add soft spring to ground. (SOL 600)
- PBUSH** Defines the nominal property values for a generalized spring-and-damper structural element.
- PBUSHT** Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
- PBUSH1D** Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
- PELAS** Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
- PELAST** Defines the frequency dependent properties for a PELAS Bulk Data entry.

### Solid Elements

- CHEXA** Defines the connections of the six-sided solid element with eight to twenty grid points.
- CPENTA** Defines the connections of a five-sided solid element with six to fifteen grid points.
- CTETRA** Defines the connections of the four-sided solid element with four to ten grid points.
- PCOMPLS** Defines the linear/nonlinear properties of an n-ply composite material laminate for a layered solid (CHEXA) element in SOL 400 and all linear solution sequences between SOL101 and SOL112.
- PLSOLID** Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
- PSLDN1** Defines the nonlinear property extensions for a PSOLID entry in SOL 400.
- PSOLID** Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

### Surface Elements

- CQUAD** Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.
- CQUAD4** Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

|                         |  |
|-------------------------|--|
| <a href="#">CQUAD8</a>  | Defines a curved quadrilateral shell or plane strain element with eight grid points.   |
| <a href="#">CQUADR</a>  | Defines an isoparametric membrane and bending quadrilateral plate element.   |
| <a href="#">CSHEAR</a>  | Defines the properties of a shear panel (CSHEAR entry).  |
| <a href="#">CTRIA3</a>  | Defines an isoparametric membrane-bending or plane strain triangular plate element.  |
| <a href="#">CTRIA6</a>  | Defines a curved triangular shell element or plane strain with six grid points.  |
| <a href="#">CTRIAR</a>  | Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element. |
| <a href="#">PCOMP</a>   | Defines the properties of an n-ply composite material laminate.  |
| <a href="#">PCOMPF</a>  | Defines the integration procedure for through the thickness integration of composite shells. (SOLs 400/600 only)   |
| <a href="#">PCOMPG</a>  | Defines global (external) ply IDs and properties for a composite material laminate.  |
| <a href="#">PLCOMP</a>  | Defines the linear/nonlinear properties of an n-ply composite material laminate of a plane stress, plane strain, or axisymmetric (CQUAD or CQUADX entry) element in SOL 400.   |
| <a href="#">PLPLANE</a> | Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.   |
| <a href="#">PSHELL</a>  | Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.   |
| <a href="#">PSHLN1</a>  | Defines nonlinear property extensions for a PSHELL or PCOMP or PCOMPG entry in SOL 400.  |
| <a href="#">PSHLN2</a>  | Defines nonlinear property extensions for a PLPLANE entry in SOL 400.  |
| <a href="#">PSHEAR</a>  | Defines the properties of a shear panel (CSHEAR entry).  |
| <a href="#">PSHEARN</a> | Defines nonlinear property extensions for a PSHEAR in SOL 400.   |
| <a href="#">SNORM</a>   | Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.   |

## Geometry

### Adaptive Meshing

|                         |   |
|-------------------------|---|
| <a href="#">HADACRI</a> | Specifies Mesh adaptivity criterion and corresponding parameters. |
| <a href="#">HADAPTL</a> | Specifies local adaptive Mesh refinement control parameters.      |

### Axisymmetry

|                        |  |
|------------------------|--|
| <a href="#">AXIC</a>   | Defines the existence of an axisymmetric conical shell problem.                  |
| <a href="#">AXIF</a>   | Defines basic parameters and the existence of an axisymmetric fluid analysis.    |
| <a href="#">AXSLOT</a> | Defines the harmonic index and the default values for acoustic analysis entries. |

|                |   |
|----------------|---|
| <b>FLSYM</b>   | Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.  |
| <b>POINTAX</b> | Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries. |
| <b>RINGAX</b>  | Defines a ring for conical shell problem.   |
| <b>SECTAX</b>  | Defines a sector of a conical shell.  |

### Coordinate Systems

|               |   |
|---------------|---|
| <b>BAROR</b>  | Defines default values for field 3 and fields 6 through 8 of the CBAR entry.  |
| <b>BEAMOR</b> | Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.   |
| <b>CORD1C</b> | Cylindrical coordinate system definition.   |
| <b>CORD1R</b> | Rectangular coordinate system definition.   |
| <b>CORD1S</b> | Spherical coordinate system definition.   |
| <b>CORD3G</b> | Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. |

### Cyclic Symmetry

|                |   |
|----------------|---|
| <b>CYAX</b>    | Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis. |
| <b>CYJOIN</b>  | Defines the boundary points of a segment in cyclic symmetry problems.           |
| <b>NLCYSYM</b> | Defines information to perform nonlinear cyclic symmetry analysis using SOL 600 |

### Fluid Points

|               |   |
|---------------|---|
| <b>ACMODL</b> | Defines modeling parameters for the Fluid-Structure Interface.  |
| <b>FREEPT</b> | Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.          |
| <b>FSLIST</b> | Defines the fluid points (RINGFL entry) that lie on a free surface boundary.  |
| <b>GRID</b>   | Defines fluid points in coupled fluid-structural analysis.  |
| <b>GRIDB</b>  | Grid point location on RINGFL.  |
| <b>GRIDF</b>  | Defines a scalar degree-of-freedom for harmonic analysis of a fluid.  |
| <b>GRIDS</b>  | Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities. |
| <b>PRESPT</b> | Defines the location of pressure points in the fluid for recovery of pressure data.   |

- RINGFL** Defines a circle (fluid point) in an axisymmetric fluid model.
- SLBDY** Defines a list of slot points that lie on an interface between an axisymmetric fluid and a set of evenly spaced radial slots.

### Grid Points

- AEGRID** Defines the location of an aerodynamic grid point.
- GRID** Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.
- GRIDB** Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.
- GRDSET** Defines default options for fields 3, 7, 8, and 9 of all GRID entries.
- SEQGP** Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

### p-Element and p-Adaptivity Analysis

- FEEDGE** Defines a finite element edge and associates it with a curve.
- FEFACE** Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
- GMBNDC** Defines a geometric boundary consisting of p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or p-solid elements.
- GMBNDS** Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.
- GMCORD** Defines a convective/follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.
- GMCURV** Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.
- GMINTC** Defines curve interface elements to connect dissimilar meshes.
- GMINTS** Defines an interface element along a surface interface between boundaries of multiple subdomains.
- GMSURF** Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
- PINTC** Defines properties for curve interface elements (see GMINTC).
- PINTS** Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.
- POINT** Define edge point for FEEDGE entry.

## Scalar Points

- EPOINT** Defines extra points for use in dynamic problems.
- SEQGP** Grid and scalar point number resequencing.
- SPOINT** Defines scalar points.

## Superelement Analysis

- CSUPER** Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.
- CSUPEXT** Assigns exterior points to a superelement.
- EXTRN** Defines a boundary connection for an external superelement.
- GRID** Defines interior points for a superelement.
- RELEASE** Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
- SEBNDRY** Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
- SEBULK** Defines superelement boundary search options and a repeated, mirrored, or collector superelement.
- SECONCT** Explicitly defines grid and scalar point connection procedures for a partitioned superelement.
- SEDLINK** Relates one design variable of a PART SE to one or more other design variables from other PART SEs.
- SEDRSP2** Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.
- SEDRSP3** Defines constituents from multiple PART SE for an external response using user-supplied routine(s).
- SEELT** Reassigns superelement boundary elements to an upstream superelement.
- SEEXCLD** Defines grid points that will be excluded during the attachment of a partitioned superelement.
- SELABEL** Defines a label or name to be printed in the superelement output headings.
- SELOC** Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.
- SEMPLN** Defines a mirror plane for mirroring a partitioned superelement.
- SEQSEP** Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.
- SESET** Defines interior grid points for a superelement.
- SETREE** Specifies superelement reduction order.

## Module Analysis

|                         |  |
|-------------------------|--|
| <a href="#">EXCLUDE</a> | Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module. |
| <a href="#">MDBCNCT</a> | Defines the Touching and Touched Contact Bodies in Different Modules.                                |
| <a href="#">MDBCTB1</a> | Defines a Contact Table for Bodies in Different Modules.   |
| <a href="#">MDBNDRY</a> | Module to Module Boundary Point Definitions.   |
| <a href="#">MDBOLT</a>  | Defines the Multi-Point Constraints for a Bolt Between Two Modules.                                  |
| <a href="#">MDBULK</a>  | Module Type Definitions.   |
| <a href="#">MDCONCT</a> | Module Boundary Point Connections.   |
| <a href="#">MDDMIG</a>  | Direct Matrix Input at Points Defined in Two or More Modules.  |
| <a href="#">MDEXCLD</a> | Module to Module Excluded Boundary Point Definitions.  |
| <a href="#">MDFAST</a>  | A Shell Patch Fastener Connection Between Two Modules.   |
| <a href="#">MDLABEL</a> | Module Output Label.   |
| <a href="#">MDLOC</a>   | Module Reposition by Translation and/or Rotation.  |
| <a href="#">MDMIR1</a>  | Defines a Module mirror by three non-collinear points on the mirror plane.                           |
| <a href="#">MDMIR2</a>  | Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane.          |
| <a href="#">MDMOVE</a>  | Defines a Module repositioning sequence. References MDMIRi, MDROTi, and MDTRAN entries.              |
| <a href="#">MDMPC</a>   | Multipoint Constraint Between Two or More Modules.   |
| <a href="#">MDMPLN</a>  | Module Reposition by Mirroring.  |
| <a href="#">MDRBE3</a>  | Rigid Body Element Between Two or More Modules, Form 2.  |
| <a href="#">MDRBE3</a>  | Interpolation Constraint Element Between Two or More Modules.  |
| <a href="#">MDRJNT</a>  | Rigid Joint Between Two Modules.   |
| <a href="#">MDROT1</a>  | Defines a Module rotation by specifying a rotation vector and reference point.                       |
| <a href="#">MDROT2</a>  | Defines a Module rotation by specifying a coordinate system axis for the rotation vector.            |
| <a href="#">MDRROD</a>  | Rigid Pin-Ended Element Connection Between Two Modules.  |
| <a href="#">MDSEAM</a>  | A Shell Patch SEAM Connection Between Two Modules.   |
| <a href="#">MDTRAN</a>  | Defines a Module translation by specifying a vector.   |
| <a href="#">MDWELD</a>  | Weld or Fastener Element Connection Between Two Modules.   |

## Loads

### Dynamic Loads

|                         |  |
|-------------------------|--|
| <a href="#">ACSRCE</a>  | Defines acoustic source as a function of power vs. frequency.  |
| <a href="#">DAREA</a>   | Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.   |
| <a href="#">DELAY</a>   | Defines the time delay term $\tau$ in the equations of the dynamic loading function.   |
| <a href="#">DLOAD</a>   | Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response. |
| <a href="#">DPHASE</a>  | Defines the phase lead term $\theta$ in the equation of the dynamic loading function.  |
| <a href="#">LOADCYH</a> | Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.   |
| <a href="#">LOADCYN</a> | Defines a physical static or dynamic load for use in cyclic symmetry analysis.   |
| <a href="#">LSEQ</a>    | Defines a sequence of static load sets.  |
| <a href="#">NOLIN1</a>  | Nonlinear transient load definition.   |
| <a href="#">NLRGAP</a>  | Defines a nonlinear transient radial (circular) gap.   |
| <a href="#">RBE3U</a>   | Defines methods to distribute applied loads to a surface. (SOL 600)  |
| <a href="#">RLOAD1</a>  | Frequency dependent excitation definition.   |
| <a href="#">TABLED1</a> | Tabular functions for generating dynamic loads.  |
| <a href="#">TLOAD1</a>  | Time dependent excitation definition.  |

### Heat Transfer Loads

|                        |  |
|------------------------|--|
| <a href="#">CONV</a>   | Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).      |
| <a href="#">CONVM</a>  | Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).    |
| <a href="#">PCONV</a>  | Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.   |
| <a href="#">PCONVM</a> | Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis. |
| <a href="#">QBDY1</a>  | Defines a uniform heat flux into CHBDYj elements.  |
| <a href="#">QBDY2</a>  | Defines grid point heat flux into CHBDYj elements.   |
| <a href="#">QBDY3</a>  | Defines a uniform heat flux load for a boundary surface.   |
| <a href="#">QHBDY</a>  | Defines a uniform heat flux into a set of grid points.   |
| <a href="#">QVECT</a>  | Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.               |
| <a href="#">QVOL</a>   | Defines a rate of volumetric heat addition in a conduction element.  |
| <a href="#">RADBC</a>  | Specifies an CHBDYi element face for application of radiation boundary conditions.   |

|                         |   |
|-------------------------|---|
| <a href="#">RADBND</a>  | Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.                           |
| <a href="#">RADCAV</a>  | Identifies the characteristics of each radiant enclosure.   |
| <a href="#">RADLST</a>  | Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.   |
| <a href="#">RADMTX</a>  | Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry. |
| <a href="#">RADSET</a>  | Specifies which radiation cavities are to be included for radiation enclosure analysis.   |
| <a href="#">SLOAD</a>   | Defines concentrated static loads on scalar or grid points.   |
| <a href="#">SLOADN1</a> | Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400   |
| <a href="#">TEMP</a>    | Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.     |
| <a href="#">TEMPB3</a>  | Defines a temperature field for the three-node beam element (CBEAM3 entry).   |
| <a href="#">TEMPBC</a>  | Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.                   |
| <a href="#">TEMPD</a>   | Specifies default initial temperature at grid points.   |
| <a href="#">TEMPN1</a>  | TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400  |
| <a href="#">VIEW</a>    | Defines radiation cavity and shadowing for radiation view factor calculations.  |
| <a href="#">VIEW3D</a>  | Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.             |

**p-element Loads**

|                        |  |
|------------------------|--|
| <a href="#">GMBC</a>   | Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.       |
| <a href="#">GMCONV</a> | Defines convection boundary conditions.  |
| <a href="#">GMLOAD</a> | Defines the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry. |
| <a href="#">GMQVOL</a> | Defines volumetric heat loads.   |
| <a href="#">TEMPF</a>  | Defines the thermal loading to be applied to a group of elements.                          |

**Static Loads**

|                        |  |
|------------------------|--|
| <a href="#">ACCEL</a>  | Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry. |
| <a href="#">ACCEL1</a> | Defines static acceleration loads at individual GRID points.   |
| <a href="#">CLOAD</a>  | Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).           |

|                 |  |
|-----------------|--|
| <b>DEFORM</b>   | Defines enforced axial deformation for one-dimensional elements for use in statics problems.   |
| <b>FORCE</b>    | Defines a static concentrated force at a grid point by specifying a vector.  |
| <b>FORCE1</b>   | Defines a static concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.  |
| <b>FORCE2</b>   | Defines a static concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.   |
| <b>FORCEAX</b>  | Defines a concentrated force on a conical shell ring.  |
| <b>FORCEi</b>   | Defines concentrated load at grid point.   |
| <b>GRAV</b>     | Defines acceleration vectors for gravity or other acceleration loading.  |
| <b>GRIDA</b>    | Defines an associative GRID point to be used in the COMBINE step of PAA.   |
| <b>LOAD</b>     | Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADB3, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries.                |
| <b>LOADCLID</b> | Defines a loading combination in PAA using the Load ID from SUBCASEs of the Parts  |
| <b>LOADCNAM</b> | Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts  |
| <b>LOADCSUB</b> | Defines a loading combination in PAA using the SUBCASE IDs from the Parts  |
| <b>LOADCYH</b>  | Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.   |
| <b>LOADCYN</b>  | Defines a physical static or dynamic load for use in cyclic symmetry analysis.   |
| <b>LOADCYT</b>  | Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.  |
| <b>LOADOF</b>   | Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.   |
| <b>LOADT</b>    | Specifies static loads that will use a table to describe their variation with pseudo-time.   |
| <b>LSEQ</b>     | Defines a sequence of static load sets.  |
| <b>MOMAX</b>    | Defines a static concentrated moment load on a ring of a conical shell.  |
| <b>MOMENT</b>   | Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.  |
| <b>MOMENTi</b>  | Defines moment at grid point.  |
| <b>PLOAD</b>    | Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.  |
| <b>PLOAD1</b>   | Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined. |

|                         |   |
|-------------------------|---|
| <a href="#">PLOAD2</a>  | Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.   |
| <a href="#">PLOAD4</a>  | Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.  |
| <a href="#">PLOADB3</a> | Defines a distributed load to a CBEAM3 element over entire length of the beam axis.   |
| <a href="#">PLOADX1</a> | Defines surface traction to be used with the CQUADX, CTRIAX, and CTRIAX6 axisymmetric element.  |
| <a href="#">PRESAX</a>  | Defines the static pressure loading on a conical shell element.   |
| <a href="#">RBE3U</a>   | Defines methods to distribute applied loads to a surface. (SOL 600)   |
| <a href="#">RFORCE</a>  | Defines a static loading condition due to an angular velocity and/or acceleration.  |
| <a href="#">SLOAD</a>   | Defines concentrated static loads on scalar or grid points.   |
| <a href="#">SPCD</a>    | Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.   |
| <a href="#">SPCR</a>    | Defines an enforced relative displacement value for a load step in SOL 400.   |
| <a href="#">TEMP</a>    | Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.   |
| <a href="#">TEMPD</a>   | Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.   |
| <a href="#">TEMPP1</a>  | Defines temperature field for surface elements.   |
| <a href="#">TEMPRB</a>  | Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery. |
| <a href="#">TEMPAX</a>  | Defines temperature sets for conical shell problems.  |

## Materials

### Anisotropic

|                      |   |
|----------------------|---|
| <a href="#">MAT2</a> | Defines the material properties for linear anisotropic materials for two-dimensional elements.  |
| <a href="#">MAT3</a> | Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries. |
| <a href="#">MAT5</a> | Defines the thermal material properties for anisotropic materials.  |
| <a href="#">MAT8</a> | Defines the material property for an orthotropic material for isoparametric shell elements.   |

|                         |  |
|-------------------------|--|
| <a href="#">MAT9</a>    | Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).  |
| <a href="#">MATORT</a>  | Define elastic 3D Orthotropic Material (SOL 400, 600. Also all linear solutions between SOL101 and 112 in conjunction with PCOMPLS, or CPYRAM).  |
| <b>Fatigue</b>          |  |
| <a href="#">MATFTG</a>  | Defines cyclic material properties (S-N + $\varepsilon$ -N) for use in fatigue analysis.   |
| <b>Fluid</b>            |  |
| <a href="#">AXIF</a>    | Includes default values for mass density and bulk modulus.   |
| <a href="#">AXSLOT</a>  | Includes default values for mass density and bulk modulus.   |
| <a href="#">CFLUIDi</a> | Includes mass density and bulk modulus.  |
| <a href="#">FSLIST</a>  | Includes mass density at free surface.   |
| <a href="#">MAT10</a>   | Defines material properties for fluid elements in coupled fluid-structural analysis.   |
| <a href="#">MFLUID</a>  | Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.  |
| <a href="#">SLBDY</a>   | Includes mass density at interface between fluid and radial slots.   |
| <b>Isotropic</b>        |  |
| <a href="#">MAT1</a>    | Defines the material properties for linear isotropic materials.  |
| <a href="#">MAT4</a>    | Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change. |
| <a href="#">MATHP</a>   | Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).  |
| <a href="#">RADM</a>    | Defines the radiation properties of a boundary element for heat transfer analysis.   |
| <b>Stress Dependent</b> |  |
| <a href="#">CREEP</a>   | Defines creep characteristics based on experimental data or known empirical creep law.   |
| <a href="#">MATS1</a>   | Specifies stress-dependent material properties for use in applications involving nonlinear materials.  |
| <a href="#">NLMOPTS</a> | Defines nonlinear material option control for SOL 400.   |
| <a href="#">TABLES1</a> | Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).                              |

## Temperature Dependent

|         |   |
|---------|---|
| MATTi   | Table references for temperature-dependent MATi materials.  |
| RADMT   | Specifies table references for temperature dependent RADM entry radiation boundary properties.  |
| TABLEM1 | Tabular functions for generating temperature-dependent material properties.   |
| TABLEST | Table references for temperature dependent MATS1 materials.   |
| TEMP    | Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.   |
| TEMPAX  | Defines temperature sets for conical shell problems.  |
| TEMPD   | Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.   |
| TEMPP1  | Defines temperature field for surface elements.   |
| TEMPRB  | Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery. |

## Miscellaneous

### Brake Squeal (SOL 400)

|         |  |
|---------|--|
| BSQUEAL | Specifies data for brake squeal analysis using Implicit Nonlinear (SOL 400). |
|---------|--|

### Comments

|    |  |
|----|--|
| \$ | Used to insert comments into the input file. Comment statements may appear anywhere within the input file. |
|----|--|

### Delete

|   |                      |
|---|----------------------|
| / | Control Input Stream |
|---|----------------------|

### Direct Matrix Input

|             |  |
|-------------|--|
| CONM1       | Defines a 6x6 mass matrix at a geometric grid point.                         |
| DMI         | Defines matrix data blocks.  |
| DMIG        | Defines direct input matrices related to grid, extra, and/or scalar points.  |
| DMIG,UACCEL | Defines rigid body accelerations in the basic coordinate system.             |
| DIMIAX      | Defines axisymmetric (fluid or structure) related direct input matrix terms. |
| TF          | Defines a dynamic transfer function.   |

**Direct Matrix Input for Aeroelasticity**

- DMIJ** Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries.
- DMIJI** Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2.
- DMIK** Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points.

**End of Input**

- ENDDATA** Designates the end of the Bulk Data Section.

**Include File**

- INCLUDE** Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.
- VCCT** Virtual Crack Closure Technique - SOL 400, SOL 600
- LORENZI** Contour Integral Approach for stress intensity calculation - SOL 600

**Materials (SOLs 400/600)**

- COHESIV** Defines data for cohesive materials. (SOL 600)
- MATDIGI** e-XstreamDigimat material interface (SOL 400)
- MATEP** Elasto-plastic material properties.
- MATF** Specifies material failure model. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS.)
- MATG** Gasket material properties.
- MATHE** Hyperelastic material properties.
- MATHED** Damage model properties for hyperelastic materials. (SOL 600)
- MATNLE** The MATNLEX entries specify advanced forms of nonlinear elastic materials. (SOL 600)
- MATORT** Elastic 3D orthotropic material properties. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS, or CPYRAM.)
- MATTEP** Thermoelastic-Plastic material properties.
- MATTF** Material Failure Model Temperature Variation
- MATTG** Temperature variation of interlaminar materials.
- MATTHE** Thermo hyperelastic material.
- MATTORT** Thermoelastic orthotropic material

|                         |   |
|-------------------------|---|
| <a href="#">MATS3</a>   | Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions. SOL 400 only.        |
| <a href="#">MATS8</a>   | Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements SOL 400 only. |
| <a href="#">MATSMA</a>  | Material properties for shape memory alloys.  |
| <a href="#">MATSORT</a> | Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials. SOL 400)                                     |
| <a href="#">MATTVE</a>  | Thermo-visco-elastic material properties  |
| <a href="#">MATVE</a>   | Viscoelastic material properties  |
| <a href="#">MATVP</a>   | Viscoplastic or creep material properties   |
| <a href="#">MCOHE</a>   | Cohesive materials (SOL 400)  |
| <a href="#">MDELAM</a>  | Defines materials for which delamination may occur.   |

### **Non-Structural Mass Distribution Selection**

|                        |  |
|------------------------|--|
| <a href="#">NSM</a>    | Non Structural Mass entry by property or element ID, value.        |
| <a href="#">NSM1</a>   | Non Structural Mass entry by property or element ID, value.        |
| <a href="#">NSMADD</a> | Non Structural Mass as sum of listed sets.                         |
| <a href="#">NSML</a>   | Lumped non structural mass entry by property or element ID, value. |
| <a href="#">NSML1</a>  | Lumped non structural mass entry by property or element ID, value. |

### **OpenFSI (SOL 400)**

|                         |   |
|-------------------------|---|
| <a href="#">FSICTRL</a> | Defines the analysis type for OpenFSI fluid structure analysis simulations.                             |
| <a href="#">WETELME</a> | Defines a wetted element used in OpenFSI fluid structure interaction simulations.                       |
| <a href="#">WETELMG</a> | Defines a wetted element used in OpenFSI fluid structure interaction simulations.                       |
| <a href="#">WETLOAD</a> | Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations. |
| <a href="#">WETSURF</a> | Defines a wetted surface used in OpenFSI fluid structure interaction simulations.                       |

### **Output Control**

|                         |   |
|-------------------------|---|
| <a href="#">BOUTPUT</a> | Defines slave nodes at which output is requested.   |
| <a href="#">CBARAO</a>  | Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output. |
| <a href="#">ECHOOFF</a> | Marks the point or points in the input file to deactivate printed echo of the Bulk Data.                      |
| <a href="#">ECHOON</a>  | Marks the point or points in the input file to activate printed echo of the Bulk Data.                        |
| <a href="#">FREEPT</a>  | Surface point location for data recovery in hydroelastic problems.  |
| <a href="#">MARPRN</a>  | Defines “print” options for SOL 600.  |

|                |  |
|----------------|--|
| <b>MLAYOUT</b> | Selects layered composite shell output to be placed in Marc's t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. (SOL 600)     |
| <b>PLOTEL</b>  | Defines a one-dimensional dummy element for use in plotting.   |
| <b>POINTAX</b> | Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. |
| <b>PRESPT</b>  | Defines the location of pressure points in the fluid for recovery of pressure data in hydroelastic problems.   |
| <b>SET1</b>    | Defines a set of grid points.  |
| <b>TSTEP</b>   | Specifies time step intervals for data recovery in transient response.   |
| <b>TSTEPNL</b> | Specifies time step intervals for data recovery in nonlinear transient response.   |

### Monitor Points

|                |   |
|----------------|---|
| <b>AECOMP</b>  | Defines a component for use in aeroelastic monitor point definition or external splines.                                |
| <b>AECOMPL</b> | Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components. |
| <b>MONPNT1</b> | Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.                        |
| <b>MONPNT2</b> | Element Monitor Output Results Item.  |
| <b>MONPNT3</b> | Sums select Grid Point Forces to a user chosen monitor point.   |
| <b>MONCNC</b>  | Stripwise aerodynamic lift and pitching moment coefficients.  |
| <b>M</b>       |   |
| <b>MONSUM</b>  | Linear combination of monitor point components.   |
| <b>MONSUM1</b> | Linear combination of two or more monitor points.   |
| <b>MONSUMT</b> | Linear combination of two or more monitor points with moment transfer.  |

### p-element Output Control

|               |   |
|---------------|---|
| <b>OUTPUT</b> | Output control for p-adaptive analysis.   |
| <b>OUTRCV</b> | Defines options for the output of displacements, stresses, and strains of p-elements. |

### Parameters

|                |   |
|----------------|---|
| <b>CAMPBLL</b> | Specifies the parameters for Campbell diagram generation.                                 |
| <b>PARAM</b>   | Specifies values for parameters used in solution sequences or user-written DMAP programs. |
| <b>MDLPRM</b>  | Specifies parameters which affect the solution of the structural model.                   |

## Solution Control

|                        |   |
|------------------------|---|
| <a href="#">ITER</a>   | Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.        |
| <a href="#">RVDOF</a>  | Degrees-of-freedom specification for residual vector computations.                  |
| <a href="#">RVDOF1</a> | Degrees-of-freedom specification for residual vector computations (alternate form). |

## Tabular Input

|                             |   |
|-----------------------------|---|
| <a href="#">DTI</a>         | Defines table data blocks.  |
| <a href="#">DTI,ESTDATA</a> | Provides override data for time and space estimation for superelement processing operations.  |
| <a href="#">DTI,INDTA</a>   | Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.   |
| <a href="#">DTI,SETREE</a>  | Defines a superelement tree that determines the superelement processing order.  |
| <a href="#">DTI,SPECSEL</a> | Correlates spectra lines specified on TABLED1 entries with damping values.  |
| <a href="#">DTI,SPSEL</a>   | Correlates output requests with frequency and damping ranges.   |
| <a href="#">DTI,UNITS</a>   | Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.  |
| <a href="#">TABDMP1</a>     | Defines modal damping as a tabular function of natural frequency.   |
| <a href="#">TABLED1</a>     | Tabular functions for generating dynamic loads.   |
| <a href="#">TABLEL1</a>     | Specifies a table of amplitude vs pseudo-time for static loads specified in LDTABL entries SOL 600 only.  |
| <a href="#">TABLEM1</a>     | Tabular functions for generating temperature-dependent material properties.   |
| <a href="#">TABLES1</a>     | Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry). |
| <a href="#">TABL3D</a>      | Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. (SOLs 400/600)   |
| <a href="#">TABLE3D</a>     | Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.   |
| <a href="#">TABLFTG</a>     | Defines cyclic load variation for use in fatigue analysis.  |
| <a href="#">TABLRPC</a>     | Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.  |
| <a href="#">TABRND1</a>     | Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.   |

### 3D Contact Region

|                         |  |
|-------------------------|--|
| <a href="#">BCBMRAD</a> | Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOLs 400/600) |
| <a href="#">BCBOX</a>   | Defines a 3D contact region.   |
| <a href="#">BCHANGE</a> | Changes definitions of contact bodies.   |
| <a href="#">BCMATL</a>  | Defines a 3D contact region by element material.   |
| <a href="#">BCMOVE</a>  | Defines movement of bodies in contact.   |
| <a href="#">BCPARA</a>  | Defines contact parameters.  |
| <a href="#">BCPROP</a>  | Defines a 3D contact region by element properties.   |
| <a href="#">BSURF</a>   | Defines a contact body or surface by element IDs.  |
| <a href="#">GMNURB</a>  | 3D contact region made up of NURBS.  |

### Old Contact Format

|                         |  |
|-------------------------|--|
| <a href="#">BCTABLE</a> | Defines a contact table.                           |
| <a href="#">BCBODY</a>  | Defines a flexible rigid contact body in 2D or 3D. |

### New Contact Format

|                         |   |
|-------------------------|---|
| <a href="#">BCTABL1</a> | Defines a contact table.  |
| <a href="#">BCONECT</a> | Defines the Touching and Touched Contact Bodies                                   |
| <a href="#">BCONPRG</a> | Geometric Contact Parameters of Touching Bodies                                   |
| <a href="#">BCONPRP</a> | Physical Contact Parameters of Touching Bodies in SOL 101 and 400                 |
| <a href="#">BCBODY1</a> | Flexible or Rigid Contact Body in 2D and 3D                                       |
| <a href="#">BCBDPRP</a> | Contact Body Parameters in SOLs 101 and 400                                       |
| <a href="#">BCRIGID</a> | Defines a Rigid Contact Body in SOLs 101 and 400                                  |
| <a href="#">BCRGSRF</a> | Rigid Contact Surface List in SOLs 101 and 400                                    |
| <a href="#">BCPATCH</a> | Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400 |
| <a href="#">BCBZIER</a> | Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400       |
| <a href="#">BCNURB2</a> | Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400              |
| <a href="#">BCNURBS</a> | Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400                 |
| <a href="#">BCTRIM</a>  | Defines the Geometry of a Trimming Curve  |

### User-Defined Subroutines and Services

|                         |   |
|-------------------------|---|
| <a href="#">BCONUDS</a> | Allows the user to provide contact routines for use with enhanced MSC Nastran contact analysis. (SOL 600 and SOL 400) |
| <a href="#">GENUDS</a>  | User Data for Notify User Defined Service or Subroutine   |

|         |  |
|---------|--|
| MATUDS  | Allows the user to provide material routines for use with enhanced MSC Nastran material models. (SOL 600 and SOL 400)                            |
| MATTUSR | Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.   |
| MATUSR  | Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only. |
| UDSESV  | Define the number and names of user state variables for material user subroutines (SOL 400)  |

## Nastran Implicit Nonlinear (SOL 600)

### 3D Contact Region (SOL 600)

|         |   |
|---------|---|
| BCBMRAD | Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOL 600) |
| SANGLE  | Defines automatic analytical contact threshold angle for multiple subcases.                                 |
| UNGLUE  | Defines grids that should be eliminated from glued contact for SOL 600 and SOL 400.                         |
| WEAR    | Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.                |

### Analysis Termination Options (SOL 600)

|        |  |
|--------|--|
| TERMIN | Control to terminate a SOL 600 analysis under certain conditions |
|--------|--|

### Bolts (SOL 600)

|         |  |
|---------|--|
| MBOLT   | Defines a bolt for use in countries outside the USA.       |
| MBOLTUS | Defines a bolt for use in the USA and all other countries. |

### Brake Squeal (SOL 600)

|       |   |
|-------|---|
| BRKSQ | Specifies data for brake squeal calculations using SOL 600. |
|-------|---|

### Composite Integration Options (SOL 600)

|         |  |
|---------|--|
| MIXTURE | Defines constituents of “composite” material on original and potentially damaged state.      |
| PCOMPF  | Defines the integration procedure for through the thickness integration of composite shells. |

## Creep Analysis (SOL 600)

- MACREEP** Controls a transient creep analysis.
- MPCREEP** Specifies input values for Marc's creep parameter when creep analysis is performed using SOL 600.
- MTCREEP** Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry.

## Element Birth and Death (SOL 600)

- ACTIVAT** This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase.
- DEACTEL** This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase.

## Elements (SOL 600)

- ALIASM** Allows selected elements which normally use a default formulation to be aliased to a different formulation.
- CSSHL** Defines a connection for a Solid Shell with 6 or 8 grid points.
- CSSHLLH** Defines conversion of CHEXA elements to Solid Shell elements.
- CSSHLM** Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements.
- CSSHLP** Defines conversion of CPENTA elements to Solid Shell elements.
- MISLAND** Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value.

## Element Properties (SOL 600)

- NTHICK** Defines nodal thickness values for beams, plates, and/or shells.
- PSSHL** Defines the properties for Solid Shell (CSSHL) elements.

## Fatigue, Fracture and Crack Propagation (SOL 600)

- LORENZI** This option gives an estimation of the J-Integral for a crack configuration using the domain integration method.
- VCCT** Virtual crack closure technique

## General Tables (SOL 600)

- MTABRV** Defines a list of tables to reverse positive and negative values and/or add points at the lower and upper end of tables.

|                         |  |
|-------------------------|--|
| <a href="#">TABD1MD</a> | Defines how TABLED1 entries are internally modified in SOL 600.  |
| <a href="#">TABL3D</a>  | Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. |

### **Heat Transfer (SOL 600)**

|                         |   |
|-------------------------|---|
| <a href="#">MPHEAT</a>  | Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.   |
| <a href="#">MTHERM</a>  | Iteration control for automatic thermal loading for structural analysis following a heat transfer analysis. |
| <a href="#">NLHEATC</a> | Defines numerical analysis parameters for SOL 600 heat transfer analysis                                    |

### **Inertia Relief (SOL 600)**

|                         |   |
|-------------------------|---|
| <a href="#">SUPORT6</a> | Inertia relief used in Nastran Implicit Nonlinear (SOL 600 only). |
|-------------------------|---|

### **Initial Conditions (SOL 600)**

|                         |  |
|-------------------------|--|
| <a href="#">IPSTRN</a>  | Defines initial plastic strain values. |
| <a href="#">ISTRESS</a> | Defines initial stress values.         |

### **Input/Output (SOL 600)**

|                          |                                |
|--------------------------|--------------------------------|
| <a href="#">MARCON</a>   | Inserts a text string in Marc. |
| <a href="#">MARCOOUT</a> | Selects data recovery output.  |

### **Matrix Input/Output (SOL 600)**

|                         |  |
|-------------------------|--|
| <a href="#">DMIGOUT</a> | Defines DMIG matrices to be output from the Marc Portion of SOL 600.   |
| <a href="#">DMIGROT</a> | Defines large rotation and other characteristics of a matrix entered using DMIG..  |
| <a href="#">MDMIAUX</a> | Specifies the DOMAINSOLVER command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only. |
| <a href="#">MDMIOUT</a> | Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600.                           |
| <a href="#">MESUPER</a> | Defines external superelement DMIG input for SOL 600 residual analyses   |
| <a href="#">MNF600</a>  | Defines auxiliary data for MSC Adams MNF files.  |

### **Solid Composites (SOL 600)**

|                        |   |
|------------------------|---|
| <a href="#">MSTACK</a> | Defines the direction in which 3D solid composites are stacked. |
|------------------------|---|

**Solution Control (SOL 600)**

|                          |   |
|--------------------------|---|
| <a href="#">MPROCS</a>   | Provides additional control for parallel processing.              |
| <a href="#">NLAUTO</a>   | Parameters for automatic load/time stepping.                      |
| <a href="#">NLHEATC</a>  | Defines Numerical Analysis Parameters for Heat Transfer Analysis. |
| <a href="#">NLSTRAT</a>  | Strategy parameters for nonlinear structural analysis.            |
| <a href="#">PARAMARC</a> | Parallel domain decomposition.                                    |
| <a href="#">RESTART</a>  | Restart data.   |

**Structural Analysis Following a Heat Transfer Analysis (SOL 600)**

|                         |   |
|-------------------------|---|
| <a href="#">MINSTAT</a> | This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file. |
|-------------------------|---|

**t16 Output Control (SOL 600)**

|                         |   |
|-------------------------|---|
| <a href="#">MT16SEL</a> | Limits elements and/or grid results to selected elements or grids for t16 and t19 file results. |
| <a href="#">MT16SPL</a> | Determines how to split a Marc t16 file into one or more smaller t16 files.                     |

**User Subroutine Control (SOL 600)**

|                         |  |
|-------------------------|--|
| <a href="#">USRSUB6</a> | Defines user subroutines used in SOL 600 only. |
|-------------------------|--|

**Explicit Nonlinear (SOL 700)****Air Bags (SOL 700)**

|                      |   |
|----------------------|---|
| <a href="#">GBAG</a> | Defines the pressure within an enclosed volume. |
|----------------------|---|

### ATB Models (SOL 700)

|                        |   |
|------------------------|---|
| <a href="#">ATBACC</a> | Defines an acceleration field that will be applied to ATB segments.   |
| <a href="#">ATBJNT</a> | This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor. |
| <a href="#">ATBSEG</a> | Defines the position and orientation of the ATB segments.   |
| <a href="#">RELEX</a>  | Defines a rigid ellipsoid whose properties and motion are defined by either ATB.  |

### Contact (SOL 700)

|                                       |   |
|---------------------------------------|---|
| <a href="#">BCBODY</a>                | Defines a flexible or rigid contact body in 2D or 3D.   |
| <a href="#">BCBODY1</a>               | Defines a flexible or rigid contact body in 2D or 3D.   |
| <a href="#">BCBOX</a>                 | Defines a 3D contact region - all elements within the region define a contact body.                               |
| <a href="#">BCELIPS</a>               | Defines a list of ellipsoid names (character strings) for use of contact analysis.                                |
| <a href="#">BCGRID (SOL 700 only)</a> | Grids to be included in contact analyses.   |
| <a href="#">BCMATL</a>                | Defines a 3D contact region by element material. All elements with the specified materials define a contact body. |
| <a href="#">BCNECT</a>                | Defines the touching and touched contact bodies used for general contact.   |
| <a href="#">BCONPRG-700</a>           | Defines geometric contact parameters of touching bodies.  |
| <a href="#">BCONPPR-700</a>           | Defines physical contact parameters of touching bodies.   |
| <a href="#">BCSEG</a>                 | Grids which are part of an element to be used in contact analyses.  |
| <a href="#">BCTABLE</a>               | Defines a contact table (old format).   |
| <a href="#">BCTABL1</a>               | Defines a contact table (new format).   |
| <a href="#">BSURF</a>                 | Defines a contact body or surface by Element IDs.   |

### Coordinate Systems (SOL 700)

|                        |   |
|------------------------|---|
| <a href="#">CORD3R</a> | Defines a moving rectangular coordinate system using three points (SOL 700 only). |
|------------------------|---|

### Dampers/Springs (SOL 700)

|                         |   |
|-------------------------|---|
| <a href="#">CDAMP1D</a> | Defines a scalar damper connection for use in SOL 700 only. |
| <a href="#">CDAMP2D</a> | Defines a scalar damper connection for use in SOL 700 only. |
| <a href="#">CELAS1D</a> | Defines a scalar spring connection for use in SOL 700 only. |
| <a href="#">CELAS2D</a> | Defines a scalar spring connection for use in SOL 700 only. |

### Direct Text Input (SOL 700)

|                         |   |
|-------------------------|---|
| <a href="#">ENDDYNA</a> | Defines the end of direct text to Dytran.   |
| <a href="#">TODYNA</a>  | Defines the start of direct text to Dytran. |

### Element Properties (SOL 700)

|                         |   |
|-------------------------|---|
| <a href="#">PBELT</a>   | Defines the properties of a belt element referenced by a CROD entry.  |
| <a href="#">PELAS1</a>  | Defines a spring property designated by a force-deflection curve for SOL 700.   |
| <a href="#">PSHELL1</a> | Defines the properties of SOL 700 shell elements that are much more complicated than the shell elements defined using the PSHELL entry. |
| <a href="#">PVISC1</a>  | Defines the properties of a nonlinear damper where the damping constant varies with the velocity.                                       |

### Eulerian Boundary (SOL 700)

|                         |  |
|-------------------------|--|
| <a href="#">BARRIER</a> | Defines a barrier for transport in a Eulerian mesh.  |
| <a href="#">FLOW</a>    | Defines the properties of a material for the boundaries of an Eulerian mesh.                             |
| <a href="#">FLOWC</a>   | Defines the properties of a material for the boundaries of a Eulerian mesh.                              |
| <a href="#">FLOWDEF</a> | Definition of default Eulerian flow boundary condition.  |
| <a href="#">FLOWT</a>   | Defines the material properties for the in- or outflow of material trough the boundary of an Euler mesh. |
| <a href="#">HYDSTAT</a> | Initializes the Euler element densities in accordance to a hydrostatic pressure profile.                 |

### Eulerian Initial Conditions (SOL 700)

|                         |   |
|-------------------------|---|
| <a href="#">CYLINDR</a> | Cylindrical shape used in the initial condition definition on the TICEUL entry.                             |
| <a href="#">SPHERE</a>  | Spherical shape used in the initial condition definition on the TICEUL entry.                               |
| <a href="#">SURFINI</a> | Defines a surface that is used for initialization of regions of an Eulerian mesh.                           |
| <a href="#">TICEL</a>   | Defines the initial values of element variables at the beginning of the analysis.                           |
| <a href="#">TICEUL1</a> | Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. |
| <a href="#">TICREG</a>  | Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. |
| <a href="#">TICVAL</a>  | Defines the initial values of an Eulerian geometric region.   |

**Eulerian Materials (SOL 700)**

|                         |   |
|-------------------------|---|
| <a href="#">EOSDEF</a>  | EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas. |
| <a href="#">EOSGAM</a>  | Defines the properties of a Gamma Law equation of state where the pressure $p$ is defined.  |
| <a href="#">EOSIG</a>   | Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives.   |
| <a href="#">EOSJWL</a>  | Defines the properties of a JWL equation of state commonly used to calculate the pressure of the detonation products of high explosives.  |
| <a href="#">EOSMG</a>   | Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure $p$ in high strain rate processes.  |
| <a href="#">EOSNA</a>   | Defines the properties of Noble-Abel equation of state.   |
| <a href="#">EOSPOL</a>  | Defines the properties of a polynomial equation of state where the pressure $p$ is defined.   |
| <a href="#">FAILJC</a>  | Defines the properties of the Johnson-Cook failure model.   |
| <a href="#">FAILMPS</a> | Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value.  |
| <a href="#">MATDEUL</a> | Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame.                |
| <a href="#">PMINC</a>   | Defines a spallation model where the minimum pressure is constant.  |
| <a href="#">SHREL</a>   | Defines an elastic shear model with a constant shear modulus.   |
| <a href="#">SHRPOL</a>  | Defines an elastic shear model with a polynomial shear modulus.   |
| <a href="#">YLDHY</a>   | Defines a yield model with zero yield stress.   |
| <a href="#">YLDJC</a>   | Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature.  |
| <a href="#">YLDMSS</a>  | Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL.   |
| <a href="#">YLDPOL</a>  | Defines a polynomial yield model where the yield stress is a function of effective plastic strain.  |
| <a href="#">YLDRPL</a>  | Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate.  |
| <a href="#">YLDSG</a>   | Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature.  |
| <a href="#">YLDTM</a>   | Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.  |

**YLDVM** Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion.

**YLDZA** Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.

### Eulerian Solid Elements (SOL 700)

**PEULER** Defines the properties of Eulerian element.

**PEULER1** Eulerian element properties.

### Euler/Lagrange Coupling (SOL 700)

**ABINFL** Defines an inflator model suited for airbag analyses.

**COUCOHF** Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface.

**COHFRIC** Allows friction and sticking during tensile conditions at the coupling surface.

**COUOPT** Defines the interaction factor and a pressure load from the covered side acting on a BSURF.

**COUP1FL** Defines the surrounding variables when a segment of a coupling surface fails.

**COUPINT** Defines the surrounding variables when a segment of a coupling surface fails.

**COUPLE** Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain.

**INFLCG** Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface.

**INFLFRC** Defines the gas fractions as a function of time for hybrid inflators.

**INFLGAS** Defines a thermically ideal gas to be used with a standard or hybrid inflator.

**INFLHB** Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface.

**INFLTNK** Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface.

**INFLTR** Defines the inflator characteristics of a COUPLE and/or GBAG subsurface.

**INITGAS** Specifies the initial gas composition inside a gasbag or Euler coupling surface.

**LEAKAGE** Defines the porosity model to be used with GBAG or COUPLE.

**PERMEAB** Defines the permeability of a COUPLE and/or GBAG (sub)surface.

**PERMGBG** Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.

**PORFCPL** Defines an interaction between two coupling surfaces through a hole.

**PORFLOW** Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface.

|         |   |
|---------|---|
| PORFLWT | Defines a time dependent flow trough a porous area of the couple surface.       |
| PORFGBG | Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG. |
| PORHOLE | Defines a hole in a COUPLE and/or GBAG surface.                                 |
| PORHYDS | Prescribes a hydrostatic pressure profile on a porous BSURF.                    |

### Hourglass Control (SOL 700)

|         |  |
|---------|--|
| HGSUPPR | Defines the hourglass suppression method and the corresponding hourglass damping coefficients. |
|---------|--|

### Initial Conditions (SOL 700)

|      |   |
|------|---|
| TIC3 | Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in Explicit Nonlinear (SOL 700) only. |
|------|---|

### Materials (SOL 700)

|        |  |
|--------|--|
| MAT1   | Defines the material properties for linear isotropic materials.  |
| MAT2   | Defines the material properties for linear anisotropic materials for two-dimensional elements.   |
| MAT8   | Defines the material property for an orthotropic material for isoparametric shell elements.  |
| MATBV  | Defines the bulk viscosity for materials.  |
| MATEP  | Elasto-plastic material properties for SOL 700 only.   |
| MATF   | Specifies failure model properties.  |
| MATFAB | Defines the properties of a bi-directional woven fabric material for shell elements.   |
| MATHE  | Specifies hyperelastic (rubber-like) material properties for nonlinear analysis.   |
| MATORT | Specifies elastic orthotropic material properties for 3-dimensional and plane strain and shell behavior for linear and nonlinear analyses. |
| MATRIG | Defines the properties of a rigid body.  |
| MATVE  | Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis                                      |

### Miscellaneous (SOL 700)

|         |  |
|---------|--|
| BIAS    | Specifies a variation of the mesh-size in one direction for use in the MESH entry.   |
| CMARKB2 | Defines a 2-noded marker beam element by means of connecting two grid points.  |
| CMARKN1 | Defines a 1-noded marker element on a grid point.  |
| DETSPH  | Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials. |

|                |   |
|----------------|---|
| <b>DYFSISW</b> | Allows activating or deactivating Fluid Structure Interaction and Eulerian solver.  |
| <b>EULFOR</b>  | Defines a body force loading (acceleration) on Euler elements per unit mass.  |
| <b>EULFOR1</b> | Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.                          |
| <b>EULFREG</b> | Defines the acceleration field for sets of Eulerian regions, The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-depended acceleration can be defined. |
| <b>FFCONTR</b> | Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling.             |
| <b>MESH</b>    | Defines a mesh.   |
| <b>PMARKER</b> | Defines the behavior of the marker element in the FV domain.  |

### Multi-Variable Parameters (SOL 700)

|                         |   |
|-------------------------|---|
| <b>DYPARAM, ATBAOUT</b> | Defines the frequency at which output is written to the main output file of ATB.  |
| <b>DYPARAM, ATBHOUT</b> | A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11.   |
| <b>DYPARAM, ATBTOUT</b> | Defines the frequency at which output is written to the time-history files of ATB.  |
| <b>DYPARAM,AUTOCOUP</b> | Defines the automatic coupling algorithm.   |
| <b>DYPARAM,AXIALSYM</b> | Axial symmetric analysis.   |
| <b>DYPARAM,AXREMAP</b>  | Allows import of a 2D axial symmetric Euler archive into a 3D simulation.   |
| <b>DYPARAM,BULKL</b>    | Defines the default value of the linear bulk viscosity coefficient.   |
| <b>DYPARAM,BULKQ</b>    | Defines the default value of the quadratic bulk viscosity coefficient.  |
| <b>DYPARAM,BULKTYP</b>  | Defines the default type of bulk viscosity.   |
| <b>DYPARAM,CFULLRIG</b> | Converts all 123456 constraints to the FULLRIG option on all entries.   |
| <b>DYPARAM,CLUFLIM</b>  | This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable. |
| <b>DYPARAM,CLUMPENR</b> | Sets the definition of the kinetic energy calculation method for Eulerian blended clumps.   |
| <b>DYPARAM,COHESION</b> | Cohesion for Coulomb Friction   |

|                  |  |
|------------------|--|
| DYPARAM,CONM2OUT | Determines if a summary of concentrated masses and their energy and momentum is written to the output file.  |
| DYPARAM,CONTACT  | Defines certain defaults for the contact definitions.  |
| DYPARAM,COSUBMXT | Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table. |
| DYPARAM,COUFRIC  | Defines the Coulomb friction scheme.   |
| DYPARAM,ELDLTH   | Print initial time step sizes for elements in the first cycle.   |
| DYPARAM,EULERCB  | Divides a Euler domain into several cubes.   |
| DYPARAM,EULTRAN  | Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver.  |
| DYPARAM,EUSUBCYC | Controls the growth of the subcycling interval in Euler computations.  |
| DYPARAM,EUSUBMAX | Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped.   |
| DYPARAM,FAILDT   | Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit.   |
| DYPARAM,FAILOUT  | Defines whether failed elements are written to the output file (ARCHIVES).   |
| DYPARAM,FASTCOUP | Defines the fast coupling algorithm.   |
| DYPARAM,FLOWMETH | Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces.  |
| DYPARAM,GEOCHECK | This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed.  |
| DYPARAM,HGCMEM   | Defines the default membrane damping coefficient for shell elements.   |
| DYPARAM,HGCOEFF  | Defines the global default hourglass damping coefficient.  |
| DYPARAM,HGCSOL   | Define the default damping coefficient for solid elements.   |
| DYPARAM,HGCTWS   | Defines the default twisting damping coefficient for shell elements.   |
| DYPARAM,HGCWRP   | Defines the default warping damping coefficient for shell elements.  |
| DYPARAM,HGSHELL  | Defines the default hourglass suppression method for shell elements.   |

|                  |  |
|------------------|--|
| DYPARAM,HGSOLID  | Defines the default hourglass suppression method for solid elements.   |
| DYPARAM,HGTYPE   | Defines the default type of hourglass suppression method.  |
| DYPARAM,HICGRAV  | Defines the value of the gravity to be used by the HIC calculations.   |
| DYPARAM,HVLFAIL  | Defines element failure on the hydrodynamic volume limit   |
| DYPARAM,HYDROBOD | Defines a body force for single hydro material in Euler.   |
| DYPARAM,IMM      | The option allows to specify the IMM method to be used.  |
| DYPARAM,INFOBJ   | Additional information about the BJOIN and spotweld connectivity will be listed in the output file.  |
| DYPARAM,INISTEP  | Defines the time step used at the start of the analysis.   |
| DYPARAM,JWLDET   | Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries. |
| DYPARAM,LIMCUB   | Defines the maximum number of cubes used to sort the grid points in a contact definition.  |
| DYPARAM,LIMITER  | Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe.   |
| DYPARAM,MATRMERG | Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly.  |
| DYPARAM,MATRMRG1 | Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties.   |
| DYPARAM,MAXSTEP  | Defines the maximum allowable time step.   |
| DYPARAM,MESHPLN  | Defines Mesh density for covering rigid planes.  |
| DYPARAM,MINSTEP  | Defines the minimum time step that causes the analysis to terminate.   |
| DYPARAM,MIXGAS   | Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature.   |
| DYPARAM,NZEROVEL | Set the velocity of a node to zero in case all attached elements have failed.  |
| DYPARAM,OLDLAGT  | Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements.  |
| DYPARAM,PARALLEL | The option allows you to gather information on the parallel section.   |
| DYPARAM,PLCOVCUT | Defines time when PLCOVER is cut off.  |
| DYPARAM,PMINFAIL | Defines Lagrangian solid element failure on reaching the spall limit.  |

|                      |   |
|----------------------|---|
| DYPARAM,RBE2INFO     | The grid points attached to MATRIG and RBE2 assemblies are listed to the output file.   |
| DYPARAM,RHOCUT       | Defines the minimum density for all Eulerian elements.  |
| DYPARAM,RJSTIFF      | Defines the stiffness of a rigid joint.   |
| DYPARAM,SHELLFRM     | Sets the default for the shell formulation for quadrilateral elements.  |
| DYPARAM,SHELMYSYS    | Defines the shell element system for the BLT shells.  |
| DYPARAM,SHPLAST      | Specifies the type of calculation used to determine the plane stress plasticity method for shells.  |
| DYPARAM,SHSTRDEF     | Specifies the default coordinate system for the stress and strain output of composite shells.   |
| DYPARAM,SHTHICK      | Specifies whether or not the thickness of the shell changes with membrane straining.  |
| DYPARAM,SLELM        | Defines whether shell sublayer variables are to be stored in the element arrays.  |
| DYPARAM,SMP,BATCHSIZ | Define batch size and number of CPU loops, per entity type.   |
| DYPARAM,SMP,CPUINFO  | Define the CPU information per entity type.   |
| DYPARAM,SNDLIM       | Defines the minimum value for the speed of sound.   |
| DYPARAM,SPHERSYM     | Enables an efficient and accurate 1D spherical symmetric solution for Eulerian materials  |
| DYPARAM,STRNOUT      | Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output.   |
| DYPARAM,TOLCHK       | To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK, write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14. |
| DYPARAM,VDAMP        | Controls the global damping in the dynamic relaxation.  |
| DYPARAM,VELMAX       | Defines the maximum velocity in Eulerian meshes.  |

|                  |   |
|------------------|---|
| DYPARAM,VELMAX1  | Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes  |
| DYPARAM,VISCPLAS | Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material. |

### Rigid Elements (SOL 700)

|         |   |
|---------|---|
| BJOIN   | Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis.                |
| RBJOINT | Defines a joint between two rigid bodies.   |
| WALL    | Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Used in Explicit Nonlinear (SOL 700) only. |

### Time Step Control (SOL 700)

|         |  |
|---------|--|
| TSTEPNL | Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. |
|---------|--|

### User Defined Subroutines (SOL 700)

|         |   |
|---------|---|
| COMPUDS | Defines an orthotropic failure model for shell composites specified by a user subroutine. |
| EOSUDS  | Defines the frequency dependent properties for an isotropic poroelastic material.         |
| FAILUDS | User defined simple failure of Eulerian materials.  |
| FLOWUDS | Defines a flow boundary on an Eulerian mesh specified by a user subroutine.               |
| FORCUDS | Defines enforced motion at grid points specified by a user subroutine.                    |
| NLOUTUD | User defined output requests for elements or Lagrangian grid points.                      |
| PORUDS  | Defines a porosity model of a COUPLE surface through a user-written subroutine.           |
| SHRUDS  | Specifies that a user subroutine is being used to define the shear modulus.               |
| TABLUDS | Specifies that a user routine is being used to define an arbitrary function.              |
| TICEUDS | User defined simple failure of Eulerian materials.  |
| YLDUDS  | Specifies that a user subroutine is being used to define a simple yield model.            |

## Solution Control

### Aerodynamic Matrix Generation

- MKAERO1** Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
- MKAERO2** Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

### Aerodynamic Parameters

- AERO** Gives basic aerodynamic parameters for unsteady aerodynamics.
- AEROS** Defines basic parameters for static aeroelasticity.

### Aeroelastic Response Analysis

- AEDW** Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG).
- AEFORCE** Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector.
- AEPARM** Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point).
- AEPRESS** Defines a vector of pressure/unit dynamic pressure associated with a particular control vector.
- AESCALE** Defines reference lengths to scale aerodynamic grid points.
- GUST** Defines a stationary vertical gust for use in aeroelastic response analysis.
- TABRNDG** Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.
- TRIM** Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.
- UXVEC** Specification of a vector of aerodynamic control point values.

### Aeroelastic Stability Analysis

- DIVERG** Defines Mach numbers (m) for a static aeroelastic divergence analysis.
- FLFACT** Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
- FLUTTER** Defines data needed to perform flutter analysis.

### Buckling Analysis

- EIGB** Defines data needed to perform buckling analysis.
- EIGRL** Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

## Cyclic Symmetry

**CYSYM** Defines parameters for cyclic symmetry analysis.

## Eigenvalue Analysis

**EIGC** Defines data needed to perform complex eigenvalue analysis.

**EIGR** Defines data needed to perform real eigenvalue analysis.

**EIGRL** Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

**EIGP** Defines poles that are used in complex eigenvalue extraction by the Determinant method.

**RVDOF,**  
**RVDOF1** Degrees-of-freedom specification for residual vector calculation.

## Monte-Carlo simulation

**MONCARL** Defines the parameters for Monte-Carlo simulations.

## Fatigue Analysis

**DTI,UNITS** Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.

**FTGDEF** Defines elements and their associated fatigue properties to be considered for fatigue analysis.

**FTGPARM** Defines parameters for a fatigue analysis

**FTGSEQ** Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or random vibration fatigue analysis using SOL 108 or SOL 111.

**FTGEVNT** Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111 by referencing FTGLOAD entries.

**FTGLOAD** Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111.

**MATFTG** Defines fatigue material properties.

**PFTG** Defines fatigue elemental properties.

**RANDPS** Defines PSDs and cross PSDs for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.

**TABLFTG** Defines tabular data for specifying fatigue cyclic loading variation for time domain fatigue analysis using SOL 101 or SOL 103.

|         |  |
|---------|--|
| TABLRPC | Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.   |
| TABRND1 | Defines tabular data for specifying PSD random vibration loading for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.  |
| TIM2PSD | Defines parameters and triggers conversion of time history channel data into corresponding Power Spectral Density (PSD) functions for use in random vibration fatigue analysis using SOL 108 or SOL 111. |
| UDNAME  | Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD.   |

### Frequency Response

|         |   |
|---------|---|
| FREQ    | Defines a set of frequencies to be used in the solution of frequency response problems. |
| FREQ1   | Defines a set of frequencies for problem solution.                                      |
| TABDMP1 | Defines modal damping as a tabular function of natural frequency.                       |

### Nonlinear Static Analysis

|         |   |
|---------|---|
| ITER    | Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153, 200 and 400.  |
| NLADAPT | Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400.                    |
| NLPARM  | Defines a set of parameters for nonlinear static analysis iteration strategy.   |
| NLPCI   | Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106).        |
| NLSTEP  | Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101. |

### Optimization (SOL 200 Only)

|         |  |
|---------|--|
| BEADVAR | Defines design region for topography (bead or stamp) optimization.   |
| BNDGRID | Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200). |
| DCONADD | Defines the design constraints for a subcase as a union of DCONSTR entries.  |
| DCONSTR | Defines design constraints.  |
| DDVAL   | Define real, discrete design variable values for discrete variable optimization.   |
| DEQATN  | Defines a design variable for design optimization.   |
| DESVAR  | Defines a design variable for design optimization.   |
| DLINK   | Relates one design variable to one or more other design variables.   |
| DOPTPRM | Overrides default values of parameters used in design optimization.  |

|         |   |
|---------|---|
| DRESP1  | Defines a set of structural responses that is used in the design either as constraints or as an objective.  |
| DRESP2  | Defines equation responses that are used in the design, either as constraints or as an objective.   |
| DRESP3  | Defines an external response using user-supplied routine.   |
| DSCREEN | Defines screening data for constraint deletion.   |
| DTABLE  | Defines a table of real constants that are used in equations (see DEQATN entry).  |
| DTABLE2 | Defines real constants from a field of property, material or connections bulk data entries which can then be invoked by a DVxREL2, DRESP2, or DRESP3 entry. |
| DVBSHAP | Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.                     |
| DVCREL1 | Defines the relation between a connectivity property and design variables.  |
| DVCREL2 | Defines the relation between a connectivity property and design variables with a user-supplied equation.  |
| DVGRID  | Defines the relationship between design variables and grid point locations.   |
| DVLREL1 | Defines the linear relation between analysis model loading and design variables   |
| DVMREL1 | Defines the relation between a material property and design variables.  |
| DVMREL2 | Defines the relation between a material property and design variables with a user-supplied equation.  |
| DVPREL1 | Defines the relation between an analysis model property and design variables.   |
| DVPREL2 | Defines the relation between an analysis model property and design variables with a user-supplied equation.   |
| DVPSURF | Design Variable to Control Surface Setting Relation   |
| DVSHAP  | Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.                                |
| MODTRAK | Specifies parameters for mode tracking in design optimization (SOL 200).  |
| SEDLINK | Multiple Design Variable Linking Across PART SE Boundary  |
| SEDRSP2 | Design Sensitivity Equation Response Quantities for PART SE   |
| SEDRSP3 | Defines External Response with User-Supplied Routines   |
| STOCHAS | Randomization of Model Parameters   |
| TOMVAR  | Defines a design region for topometry optimization (element-by-element optimization).   |
| TOPVAR  | Defines a topology design region for topology optimization.   |

### p-element and p-Adaptivity Analysis

- ADAPT** Defines controls for p-version adaptive analysis.
- PSET** Describes polynomial order distribution and is selected by the ADAPT Case Control command.
- PVAL** Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

### Random Response

- RANDPS** Defines load set power spectral density factors for use in random analysis.
- RANDT1** Defines time lag constants for use in random analysis autocorrelation function calculation.
- RCROSS** Cross-power spectral density and cross-correlation function output.
- TABRND1** Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

### Rotordynamics

- RGYRO** Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.
- ROTHYBD** Hybrid damping for rotors
- ROTOR** Rotor Model Definition
- ROTORAX** Axisymmetric Model Rotor Definition
- ROTORG** Specifies grids that compose the rotor line model.
- ROTORSE** Specifies grids that compose the rotor line model.
- RSPINR** Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.
- RSPINT** Specifies rotor spin rates for nonlinear transient analysis.
- UNBALNC** Specifies an unbalanced load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

### Transient Response

- TIC** Defines values for the initial conditions of variables used in structural transient analysis.
- TSTEP** Defines time step intervals at which a solution will be generated and output in transient analysis.
- TSTEPNL** Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, and 99.

## Entries A - B

| \$ | Comment |
|----|---------|
|----|---------|

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

**Format:**

\$ followed by any characters out to column 80.

**Example:**

```
$ TEST FIXTURE-THIRD MODE
```

**Remarks:**

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.
3. For free field entries (See Format of Bulk Data Entries above), a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank.
4. For the default NASTRAN SYSTEM (IFPSTAR) = YES, input using FIXED field, a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank.  
The OLD IFP (NASTRAN SYSTEM (IFPSTAR) = NO) will fail with this option.

/

## Control Input Stream

Allows the user control over field width, and on restart entry deletion, insert and update, and process control.  
(This entry's execution is dependent on whether NASTRAN SYSTEM(444)= 0 or 1.)

Format:

| 1 | 2                                     | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---------------------------------------|---|---|---|---|---|---|---|----|
| / | Alpha numeric text as described below |   |   |   |   |   |   |   |    |

Examples:

/ 0 12

/delete all

/ Delete FORCE1,PARAM MATVP

| Descriptor   | Meaning   |
|--|---|
| /  | Must be first character and occur somewhere in fields one through 8. (Character)  |
| <i>If NASTRAN SYSTEM(444)=0, then the n1, n2 directive below is the only valid form:</i>                               |   |
| n1,n2  | Two integers indicating following number of lines of previous input are to be deleted.<br>(Integer, n1 ≥ 10, n2 > n1). /, n1,n2 directive must be entered before any new data is added to the input stream.   |
| <i>If NASTRAN SYSTEM(444)=1, then the above n1, n2 directive will not work and only the following forms are valid:</i> |   |
| DELETE   | Keyword indicating following listed items are to be deleted. (Character) /DELETE directive must be entered before any new data is added to the input stream.  |
| ALL  | Keyword indicating following a /DELETE directive. All previous input items will be deleted on RESTART. (Character)  |
| <i>List of names</i>   | A list of Bulk Data names to be deleted following a /DELETE directive. All previous input Bulk entries with the names indicated will be deleted on RESTART. (Character)   |
| SFW  | A Keyword that is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9. See Remark 1. (Character)  |
| n  | Single Field Width value n used with SFW. This is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9 to the value n.(Integer 8 ≤ n ≤ 14, Default = 8). If double field is selected by use of the * option, then Field 2 thru Field 9 are of 2n width. (Field 1 and Field 10 always remain of width 8). See Remark 1. |

| Descriptor | Meaning  |
|------------|--|
| SKIPON     | Input data following this keyword would not be processed in the current execution. (Character). This command is terminated by one of three methods: 1) The occurrence of a SKIPOFF. 2) The occurrence of a BEGIN (Such as BEGIN SUPER=10) 3) The occurrence of an ENDDATA. |
| SKIPOFF    | Keyword turning off the SKIPON command. (Character)  |
| INSERT     | Keyword that implies that entries when placed into the input stream must not be previously present. See Remark 1. (Character; Default for INITIAL runs.)   |
| UPDATE     | Keyword that implies that entries when placed into the input stream are new or are to replace existing entries. See Remark 1. (Character; Default for RESTART runs.)   |
| DUPMETH    | Keyword that allows reversion to the original IFP, DUPTOL tolerance selection. /DUPMETH=OLD selects original IFP tolerance selection. See the DPBLKTOL(402) system cell description and Remark 1. (Character; Default NEW for IFPSTAR tolerance selection).                |
| DUPTOL     | Keyword that allows a closeness tolerance value when searching for removal of duplicate entries. (Character)   |
| v          | Tolerance value, used with DUPTOL, when searching for removal of duplicate entries. See Remark 1. (Real $\geq 0$ ; Default 1.0-5.)   |
| LISTON     | Keyword to indicate a "list of data to follow" See Remark 10. (Character)  |
| LISTOFF    | Keyword to indicate a "list of data is terminated" See Remark 10. (Character)  |

### Remarks:

1. If the SFW, INSERT, UPDATE, DUPMETH, DUPTOL Keywords and any associated value (n, v) occur in the first BEGIN BULK Section the values set will carry forward to any other BEGIN section included. If a user wishes to have some later BEGIN have different control values, then the BEGIN BULK values can be overridden for that particular BEGIN Section by the inclusion of / entries particular to that specific BEGIN Section. The DUPMETH must come before the DUPTOL entry. Currently IFPSTAR will not recognize the settings from the command line or RC file due to default usage issues at the application state and real value passing through the component level. When DUPMETH is OLD and DPBLKTOL or DUPTOL is zero or positive, remove grids with duplicated id without checking CP, CD, PS ,SE fields and distance tolerance.

### Example:

```
/DUPMETH=OLD
/DUPTOL=0.1
```

With the following NASTRAN entry required:  
NASTRAN SYSTEM(402)=0.1

2. The input stream into Nastran is now part of a Common Data Model component and entries are committed to a data base as soon as they are interpreted in a single unsorted pass. When the user requests a sorted ECHO, the ECHO represents a binary value converted back into a column character location. It does not represent the actual binary entry value! Unsorted ECHO respects the request list of the ECHO command.
3. If an improperly formatted bulk data entry is encountered, the true file line number of the error is printed along with reference to the Parent entry. If the offending entry is in an include file the file is identified.
4. Usage of the continuation mnemonic is no longer needed and its usage is not recommended.
5. Default Fixed Fields are 80 characters (see SFW). Free field supports 128 characters.
6. Because the Common Data Model component entries are committed to a data base as soon as they are interpreted in a single unsorted pass, the /DELETE syntax is now required to be used on restart.
7. If an /INSERT is present on RESTART the Common Data Model component will signal an error condition EXCEPT when certain objects such as GRID entries have been classified as Remove Duplicate or when objects such as SPC that allow superposition are encountered.
8. Entries that allow superposition are always in /INSERT.
9. If NASTRAN SYSTEM(444)=1, then the Nastran Replication feature is not recommended.
10. For convenience in entering an ordered list of elements (tuples) data or, such as table value inputs, the following is allowed for example:

tabled1,111

/liston

1.0 1.0

2.0 1.0

3.0 2.0

4.0 2.0

5.0 1.0

6.0 1.0

/listoff

,endt

or for example if user likes commas:

tabled1,112

/liston

1.0, 1.0

2.0, 1.0

3.0, 2.0

4.0, 2.0

5.0, 1.0  
6.0, 1.0  
*/listoff*  
,endt

**ABINFL**

## Inflator Model to be Used With GBAG or COUPLE Entries

Defines an inflator model suited for airbag analyses. The inflator model is defined as part of the GBAG or COUPLE surface. Used in SOL 700 only.

Format:

| 1      | 2   | 3     | 4     | 5       | 6        | 7     | 8      | 9 | 10 |
|--------|-----|-------|-------|---------|----------|-------|--------|---|----|
| ABINFL | CID | INFID | SUBID | INFTYPE | INFTYPID | COEFF | COEFFV |   |    |

Example:

|        |     |   |     |        |    |  |       |  |  |
|--------|-----|---|-----|--------|----|--|-------|--|--|
| ABINFL | 201 | 1 | 120 | INFLHB | 11 |  | 0.012 |  |  |
|--------|-----|---|-----|--------|----|--|-------|--|--|

| Descriptor | Meaning  |
|------------|--|
| CID        | Unique number of a ABINFL entry. (Integer > 0; Required)   |
| INFID      | Number of a set of ABINFL entries NFID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)                                     |
| SUBID      | Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the as defined on the GBAG or COUPLE entry. (Integer > 0; Required)     |
| INFTYPE    | Defines the type of inflator. (Character; Required)  |
|            | INFLTR      The INFLTR logic is used to model inflators in an air bag.   |
|            | INFLHB      The INFLHB logic is used to model hybrid inflators in an air bag.  |
|            | INFLCG      The INFLCG logic models a cold gas inflator.   |
|            | INFLTNK      The INFLTNK logic models the inflator properties (mass flow rate and inflator gas temperature) calculated from the empirical results. |
| INFTYPID   | Number of the entry selected under INFTYPE, for example, INFLTR,INFTYPID. (Integer > 0; Required)  |
| COEFF      | Method of defining the area coefficient. (Character, CONSTANT)   |
|            | CONSTANT      The area coefficient is constant and specified on COEFFV.  |
|            | TABLE      The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.                          |
| COEFFV     | The area coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or 1 > 0)                                    |

Remarks:

1. The INFLTR or INFLHB inflator geometry and location is defined by a BSURF, BCBOX, BCPROP, BCMATL or BCSEG. The area of the hole through which the gas enters is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete subsurface area, while a value of COEFFV = 0.0 will result in a closed inflator area with no inflow.

2. This allows for setting up the exact same model for either a uniform pressure model or an Euler Coupled model. This makes it possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

**ABSNMVB**

Meta data group Boolean name/value pairs

Associate Boolean Name/Value pairs to meta data collectors (ABSTRACT)

Format:

| 1       | 2      | 3 | 4     | 5    | 6     | 7    | 8      | 9   | 10 |
|---------|--------|---|-------|------|-------|------|--------|-----|----|
| ABSNMVB | ABSTID |   | NAME1 | VAL1 | NAME2 | VAL2 | -etc.- | ... |    |

Example:

| 1       | 2  | 3 | 4        | 5    | 6       | 7     | 8 | 9 | 10 |
|---------|----|---|----------|------|---------|-------|---|---|----|
| ABSNMVB | 50 |   | STIFFNER | TRUE | VARSECT | FALSE |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| ABSTID     | Unique identification number for meta data group (ABSTRACT). (Integer>0).      |
| NAMEi      | Up to eight characters defining the name of this Name/Value pair. (Character). |
| VALi       | Value of this Name/Value pair -Boolean- data (Character) TRUE or FALSE.        |

Remarks:

1. This entry has no impact on the solution.
2. The only allowed character values are “TRUE” or “FALSE”.

**ABSNMVI**

Meta data group Integer name/value pairs

Associate Integer Name/Value pairs to meta data collectors (ABSTRACT).

**Format:**

| 1       | 2      | 3 | 4     | 5    | 6     | 7    | 8      | 9 | 10 |
|---------|--------|---|-------|------|-------|------|--------|---|----|
| ABSNMVI | ABSTID |   | NAME1 | VAL1 | NAME2 | VAL2 | -etc.- |   |    |

**Example:**

| 1       | 2  | 3 | 4     | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|----|---|-------|----|---|---|---|---|----|
| ABSNMVI | 50 |   | SPANS | 12 |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| ABSTID     | Unique identification number for meta data group (ABSTRACT). (Integer>0).      |
| NAMEi      | Up to eight characters defining the name of this Name/Value pair. (Character). |
| VALi       | Value of this Name/Value pair (Integer).                                       |

**Remarks:**

1. This entry has no impact on the solution.

**ABSNMVR**

Meta data group Real Name/Value pairs

Associate Real Name/Value pairs to meta data collectors (ABSTRACT).

**Format:**

| 1       | 2      | 3 | 4     | 5    | 6     | 7    | 8      | 9 | 10 |
|---------|--------|---|-------|------|-------|------|--------|---|----|
| ABSNMVR | ABSTID |   | NAME1 | VAL1 | NAME2 | VAL2 | -etc.- |   |    |

**Example:**

| 1       | 2  | 3 | 4      | 5    | 6      | 7    | 8 | 9 | 10 |
|---------|----|---|--------|------|--------|------|---|---|----|
| ABSNMVR | 50 |   | ANGLEA | 15.0 | ANGLEB | 45.0 |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| ABSTID     | Unique identification number for meta data group (ABSTRACT). (Integer>0)      |
| NAMEi      | Up to eight characters defining the name of this Name/Value pair. (Character) |
| VALi       | Value of this Name/Value pair (Real)  |

**Remarks:**

1. This entry has no impact on the solution.

**ABSNMVS**

## Meta data group String name/value pairs

Associate String Name/Value pairs with meta data collectors (ABSTRACT).

**Format:**

| 1       | 2      | 3 | 4 | 5 | 6    | 7 | 8 | 9 | 10 |
|---------|--------|---|---|---|------|---|---|---|----|
| ABSNMVS | ABSTID |   |   |   |      |   |   |   |    |
|         | NAME1  |   |   |   | VAL1 |   |   |   |    |
|         | NAME2  |   |   |   | VAL2 |   |   |   |    |
|         | -etc.- |   |   |   |      |   |   |   |    |

**Example:**

| 1       | 2       | 3                                  | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|---------|------------------------------------|---|---|---|---|---|---|----|
| ABSNMVS | 50      |                                    |   |   |   |   |   |   |    |
|         | ABSNAME | B pillar model year 2020           |   |   |   |   |   |   |    |
|         | ABSDESC | B pillar including locking bushing |   |   |   |   |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| ABSTID     | Unique identification number for meta data group (ABSTRACT). (Integer>0)  |
| NAMEi      | Up to eight characters defining the name of this Name/Value pair. (Character).  |
| VALi       | Value of this Name/Value pair. May consist of any Character A-Z, or numbers, and underscore or dash. Limited to 56 characters. (Character). |

**Remarks:**

1. This entry has no impact on the solution.
2. The name/value pair entries may be specified in any order.

3. In the HDF5 viewer

The screenshot shows the HDF5 viewer interface with two main sections:

- INPUT** section:
  - ABSTRACT** dataset:
    - ABSNMVS**
    - IDENTITY: ABSTID: ID**
    - NAMEVAL: Position value: "ABSNAME", "ABSDESC"**
- ABSTRACT** section:
  - IDENTITY: ABSTMN: ID, NASTNM, NASTIDs**
  - NASTIDs: NASTNM IDs**

Below these sections, the HDF5 viewer interface displays the following data structures:

- Object Attribute Info** and **General Object Info** panes.
- Attribute Creation Order** and **Creation Order Number** panes.
- Number of attributes = 1** pane.
- ABSTRACT** dataset:
  - ABSNMVS** attribute (Type: Array, Size: 1, Value: 0)
  - IDENTITY** attribute
  - NAMEVAL** attribute
- ABSTRACT** dataset:
  - IDENTITY** attribute
  - NASTIDs** attribute
- CONSTRAINT** dataset
- DOMAINS** dataset
- ELEM** dataset
- RBD** dataset

Three tables are shown on the right side of the viewer:

- NAME SVAL** table:
 

| NAME       | SVAL                               |
|------------|------------------------------------|
| 0 ABSNAME  | RIGHT SPAN                         |
| 1 ABSNAME  | BEAM SHAPE 1                       |
| 2 ABSNAME  | LEFT SPAN                          |
| 3 ABSNAME  | POINT MASS IN HOLE                 |
| 4 ABSDESC  | THIS IS A REMOTE POINT MASS IN HO. |
| 5 ABSNAME  | REGIONS APPLICATION/REGION PON.    |
| 6 ABSNAME  | FIXED BASE CONSTRAINT              |
| 7 ABSDESEL | WOLD                               |
| 8 ABSNAME  | FIXED                              |
- ABSTNAME ABSTID NASTNAME\_NASTIDS\_NASTIDS\_POS NASTIDS** table:
 

| ABSTNAME      | ABSTID | NASTNAME | NASTIDS_Pos | NASTIDS |
|---------------|--------|----------|-------------|---------|
| 3 BEAMSPAN 3  | CBEAM  | 50       | 48          | 1       |
| 4 CONSTRAIN 7 | SPC    | 98       | 1           | 1       |
| 5 LEVENT 8    | PLOAD  | 99       | 1           | 1       |
| 6 LEVENT 10   | FORCE  | 100      | 1           | 1       |
| 7 LOAD 9      | PLOAD  | 101      | 1           | 1       |
| 8 LOAD 11     | FORCE  | 102      | 1           | 1       |
| 9 MASS 1      | CONN2  | 103      | 1           | 1       |
| 10 MASS 1     | GRID   | 104      | 1           | 1       |
| 11 MASS 1     | RBE3   | 105      | 1           | 1       |
- NASTID** table:
 

| NASTID    |
|-----------|
| 97 2550   |
| 98 5      |
| 99 12     |
| 100 13    |
| 101 12    |
| 102 13    |
| 103 2857  |
| 104 13828 |
| 105 7589  |
| 106 13828 |

Arrows from the viewer interface point to specific fields in the tables, indicating their correspondence to the attributes defined in the datasets.

4. In the above Example, the following SET3 entries might include:

```
$ CBEAM elements for B pillar
SET3, 10, ELEM, 7701, THRU, 7799
$ CBUSH elements for locking
SET3, 30, ELEM, 66, 67, 68, 69
$ RIGID ELEMENTS
SET3, 90, RBEIN, 9991, THRU, 9996
```

**ABSTRACT****Meta Data Collector for HDF5 output and post processing**

Allows the user to define an arbitrary group of Nastran entities (ABSTRACTion), using any Nastran keyword type including elements, grids, materials, etc. and associate a name and description for the group.

**Format:**

| 1        | 2      | 3      | 4      | 5       | 6       | 7       | 8      | 9 | 10 |
|----------|--------|--------|--------|---------|---------|---------|--------|---|----|
| ABSTRACT | ABSTNM | ABSTID | NASTNM | NASTID1 | NASTID2 | NASTID3 | -etc.- |   |    |

**Example:**

| 1        | 2      | 3  | 4     | 5  | 6 | 7 | 8 | 9 | 10 |
|----------|--------|----|-------|----|---|---|---|---|----|
| ABSTRACT | PILLAR | 50 | CBEAM | 10 |   |   |   |   |    |
| ABSTRACT | PILLAR | 50 | CBUSH | 30 |   |   |   |   |    |
| ABSTRACT | PILLAR | 50 | RBE3  | 90 |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| ABSTNM     | Up to eight characters for the name of the meta data group. (Character).                                   |
| ABSTID     | Unique identification number for the meta data group. (Integer>0).   |
| NASTNM     | Up to eight characters for the name of the Nastran entity being added to the meta data group. (Character). |
| NASTIDi    | ID referencing a Nastran entity of the specified type or a set of such entities.                           |

**Remarks:**

1. The group definition is transmitted from the input file to the .op2 and/or .h5 results file with no impact on the solution.
2. The group definition can subsequently be recovered from either the .op2 or .h5 file by a post-processor.
3. Associated with this entry are the entries: [ABSNMVS](#), [ABSNMVB](#), [ABSNMVI](#), [ABSNMVR](#).

**ACCEL****Acceleration Load**

Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.

**Format:**

| 1     | 2    | 3    | 4    | 5    | 6                        | 7   | 8 | 9 | 10 |  |
|-------|------|------|------|------|--------------------------|-----|---|---|----|--|
| ACCEL | SID  | CID  | N1   | N2   | N3                       | DIR |   |   |    |  |
|       | LOC1 | VAL1 | LOC2 | VAL2 | Continues in Groups of 2 |     |   |   |    |  |

**Example(s):**

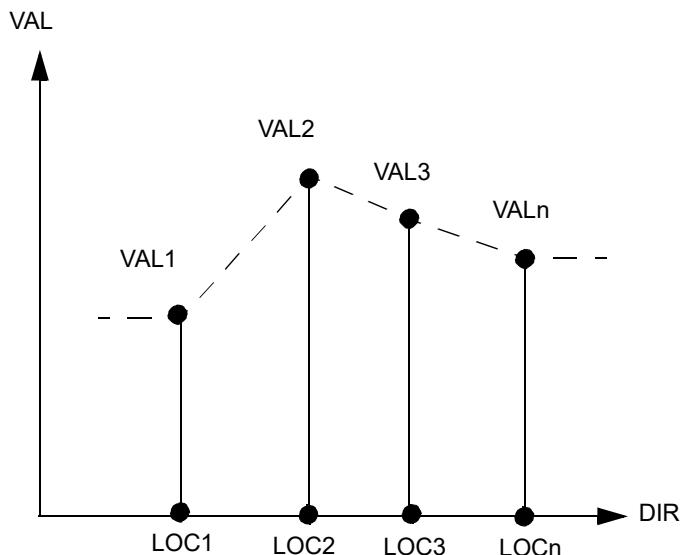
|       |     |     |        |     |     |   |  |  |  |
|-------|-----|-----|--------|-----|-----|---|--|--|--|
| ACCEL | 100 | 2   | 0.0    | 1.0 | 2.0 | X |  |  |  |
|       | 0.0 | 1.0 | 1000.0 | 3.0 |     |   |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number (Integer > 0)  |
| CID        | Coordinate system identification number. (Integer > 0; Default = 0)                                     |
| Ni         | Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0 ) |
| DIR        | Component direction of acceleration variation. (Character; one of X, Y, and Z)                          |
| LOCi       | Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)  |
| VALi       | The load scale factor associated with location LOCi. (Real)   |

**Remarks:**

- For all grids of the model, the acceleration vector is defined by  $\ddot{\mathbf{a}} = VAL \cdot \vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\ddot{\mathbf{a}}$  is equal to VAL times the magnitude of  $\vec{N}$ . The scale factor VAL for each grid is found linearly interpolating the DIR coordinate of the grid between table values LOCi/VALi. If the GRID point coordinate in coordinate system CID is outside the range of the table, VAL is determined either from VAL1 or VALn (the last value, see the following figure).
- This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL1 loads, by specification on a LOAD entry. The SID on an ACCEL entry may not be the same as that of any other load entry.
- This acceleration load does not include effects due to mass on scalar points.
- A CID of zero references the basic coordinate system.
- The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.

6. A minimum of two pairs of {LOC<sub>i</sub>, VAL<sub>i</sub>} data must be specified.



Definition of Load Scale Factor vs Location

7. If Modules are present then this entry may only be specified in the main Bulk Data section.
8. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

**ACCEL1****Acceleration Load**

Defines static acceleration loads at individual GRID points.

**Format:**

| 1      | 2       | 3       | 4      | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|---------|---------|--------|----|----|----|---|---|----|
| ACCEL1 | SID     | CID     | A      | N1 | N2 | N3 |   |   |    |
|        | GRIDID1 | GRIDID2 | -etc.- |    |    |    |   |   |    |

**Example(s):**

|        |     |    |      |     |      |     |    |    |  |
|--------|-----|----|------|-----|------|-----|----|----|--|
| ACCEL1 | 100 | 2  | 10.0 | 1.0 | 2.0  | 0.0 |    |    |  |
|        | 1   | 2  | 3    | 4   | THRU | 10  | BY | 2  |  |
|        | 20  | 21 | THRU | 30  | 40   | 52  | 69 | 70 |  |
|        | 82  | 90 | 100  |     |      |     |    |    |  |

| Descriptor               | Meaning   |
|--------------------------|---|
| SID                      | Load set identification number (Integer > 0)  |
| CID                      | Coordinate system identification number. (Integer > 0; Default = 0)   |
| A                        | Acceleration vector scale factor. (Real)  |
| Ni                       | Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0 )                           |
| GRIDID <sub>i</sub> LIST | List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer > 0) |

**Remarks:**

1. The acceleration vector is defined by  $\ddot{a} = A \cdot \vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\ddot{a}$  is equal to  $A$  times the magnitude of  $\vec{N}$ .
2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL loads, by specification on a LOAD entry. The SID on an ACCEL1 entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.
5. ACCEL1 loads are not supported with super elements.
6. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

## ACCSSPT

## Access Point Definition

Defines a grid interior to an external superelement whose motion will be retained in the residual structure of the assembled superelement model. This entry is used during the external superelement creation run and is valid only in external superelements.

Format:

| 1       | 2    | 3    | 4    | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|------|------|------|-----|---|---|---|---|----|
| ACCSSPT | GID1 | GID2 | GIDi | etc |   |   |   |   |    |

Example:

|         |     |     |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|
| ACCSSPT | 117 | 119 |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| GIDi       | Identification number of a grid point designated as an access point (Integer>0) |

Remarks:

1. An access point may only be defined in an external superelement.
2. Access point IDs must be unique across all external superelements.
3. Access points must be interior points in the external superelement.
4. Access points will be automatically placed in the m-set of the residual structure of the assembly and will have the same displacements in the residual structure as it does in the external superelement.
5. If the user needs the access points to be in the a-set, or if user needs to connect access points from different superelements via MPCs, the following two parameters are required:  
PARAM, SEP1XOVR, 128  
PARAM, AUTOMSET, YES
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**ACLOAD**

## ACTRAN Acoustic Pressure Load Matrices for SOL 108/111

Defines ACTRAN acoustic pressure load matrices.

**Format:**

| 1      | 2   | 3     | 4     | 5    | 6    | 7     | 8 | 9 | 10 |
|--------|-----|-------|-------|------|------|-------|---|---|----|
| ACLOAD | SID | UNIT1 | UNIT2 | SCLR | SCLI | LSQID |   |   |    |

**Example(s):**

|        |     |    |    |     |  |  |  |  |  |
|--------|-----|----|----|-----|--|--|--|--|--|
| ACLOAD | 101 | 41 | 42 | 1.5 |  |  |  |  |  |
|--------|-----|----|----|-----|--|--|--|--|--|

| Field | Contests  |
|-------|---|
| SID   | Set identification number. See Remark 1. (Integer > 0)  |
| UNIT1 | Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)                               |
| UNIT2 | Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)                         |
| SCLR  | The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)          |
| SCLI  | The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)     |
| LSQID | Load SeQuence IDentification number on an ACTRAN file with multiple load cases. (Integer > 0, default=1). |

**Remarks:**

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID for frequency response analysis.
2. The following type of ASSIGN should be specified in the FMS section with the vacant unit number (see ASSIGN statements). The unit number cannot be selected doubly.  

```
ASSIGN INPUTT2='ACTRAN_pressure.f70' UNIT=41
ASSIGN INPUTT4='ACTRAN_pressure.op4' UNIT=42
```
3. SID must be unique for all RLOAD1, RLOAD2, ACSRCE and ACLOAD entries.
4. Refer the ACTRAN manual for the details of exportation of the acoustic pressure load matrix data for Nastran.
5. The residual vectors for ACLOAD will not be computed.
6. Value of LSQID that is greater than the actual load cases in the ACTRAN file is a fatal error.

**ACMODL****Fluid-Structure Interface Parameters**

Optional entry to define modeling parameters for Fluid-Structure Interface.

**Format:**

(If METHOD=“BW”)

| 1      | 2     | 3       | 4        | 5    | 6      | 7      | 8      | 9       | 10 |
|--------|-------|---------|----------|------|--------|--------|--------|---------|----|
| ACMODL | INTER | INFOR   | FSET     | SSET | NORMAL | METHOD | SKNEPS | DSKNEPS |    |
|        | INTOL | ALLSSET | SRCHUNIT |      |        |        |        |         |    |

(If METHOD=“CP”)

|        |       |       |      |      |        |        |  |  |  |
|--------|-------|-------|------|------|--------|--------|--|--|--|
| ACMODL | INTER | INFOR | FSET | SSET | NORMAL | METHOD |  |  |  |
|--------|-------|-------|------|------|--------|--------|--|--|--|

**Example(s):**

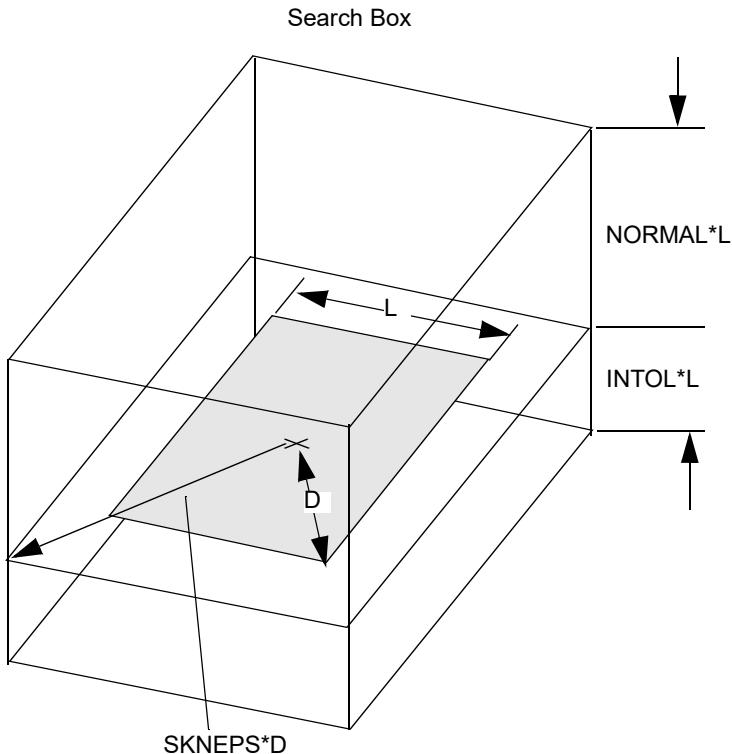
|        |       |  |  |  |        |  |  |  |  |
|--------|-------|--|--|--|--------|--|--|--|--|
| ACMODL | IDENT |  |  |  | NORMAL |  |  |  |  |
|--------|-------|--|--|--|--------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| INTER      | Type of structure-fluid interface. (Character = “IDENT” or “DIFF”; Default = “DIFF”)   |
| INFOR      | For METHOD = “BW” and INTER = “DIFF”, indicates if FSET and SSET are used to define the fluid-structure interface, “NONE” if not used, and whether they contain grids or elements. (Character = “GRIDS”, “ELEMENTS”, “ALL”, or “NONE”, Default = “NONE”) |
| FSET       | For METHOD = “CP” and INTER = “DIFF”, indicates if FSET and SSET are used to define the fluid-structure interface, “NONE” if not used. See Remark 10. (Character = “ALL”, or “NONE”, Default = “NONE”)   |
| SSET       | Optional identification of a SET1 entry that contains a list of fluid elements or grids on the fluid “skin”. See Remark 2. (Integer > 0 or blank)  |
| NORMAL     | Optional identification of a SET1 entry that contains a list of structural elements or grids on the structure-fluid interface. See Remark 2. (Integer > 0 or blank)  |
| METHOD     | Fluid normal tolerance. See Remark 5. (Real; Default = 1.0 (Real; .001 for IDENT)<br>Default = “BW”<br>“BW” = Body in White method<br>“CP” = Closed Pressure Vessel Method<br>See Remarks 10. and 11.)   |
| SKNEPS     | Fluid skin growth tolerance. (Real; Default 0.5)   |
| DSKNEPS    | Secondary fluid skin growth tolerance (Real; Default .75)  |
| INTOL      | Tolerance of inward normal. (Real; Default .5)   |

| Descriptor | Meaning  |
|------------|--|
| ALLSSET    | If "NO" then SSET structure is searched and coupled if found. If 'YES' then all the structure given by SSET is coupled. (Character = 'YES', or 'NO'; Default = 'NO') |
| SRCHUNIT   | Search units. (Character='ABS' for absolute model units or 'REL' for relative model units based on element size; Default = 'REL')                                    |

**Remarks:**

1. Only one ACMODL entry is allowed. In general, for large irregular models, it is recommended that, initially, this entry not be used, so the defaults will be applied.
2. For METHOD = "BW" and INTER = "DIFF" (Default), FSET and SSET refer to either grids or elements as selected below. For INTER = "IDENT", FSET and SSET refer to grids.
  - a. For INTER = "DIFF", INFOR = "ELEMENTS", for FSET, the search algorithm is restricted to elements referenced by FSET. For SSET, the search algorithm is restricted to elements referenced by SSET. This allows the user to de-select specific structural faces of a solid structural element. Both these sets are optional and the user can have one without the other.
  - b. For INTER = "DIFF", INFOR = "GRIDS", for FSET, the search algorithm is restricted to grids referenced by FSET. This allows the user to deselect fluid grids. Fluid grid selection is the only way to deselect specific fluid faces. For SSET, the search algorithm is restricted to grids referenced by SSET. This allows the user to de-select structural grids. Both these sets are optional and the user can have one without the other.
  - c. For INTER = "IDENT", INFOR = "ALL", the points referenced by FSET and SSET must lie exactly on the fluid-structure interface. These sets are optional, but if used, both must be present or no fluid interface is calculated.
3. For INTER = "DIFF", a .PCH file is created with a SET1 representing the fluid "skin" and a SET1 representing the structure interface. This file is useful for graphic post-processing for viewing the interface. It also produces the sets that can be used as FSET and SSET.
4. For ALLSSET = 'NO' (default) the elements and grids determined by the couplings algorithm are written to the .PCH file. The user can then deselect elements or grids as defined by the .PCH file by editing them out of the SET1 entries defined in the file and referencing the edited SET1 with the SSET. To add structural elements that the coupling algorithm did not include in the .PCH file, it is not sufficient to just include them on the SET1 entry referenced by SSET. In addition, ALLSSET = 'YES' must be specified.
5. NORMAL determines the height of the fluid box in the outward normal direction to the fluid surface. The fluid box is used to locate the structural elements used in defining the fluid-structure coupling matrix. If L is the smallest edge of the fluid element surface, then the height of the box is L x NORMAL.  
For INTER = "IDENT", NORMAL = .001 is the default and represents a tolerance, in units of length, used in determining the fluid-structure interface.
6. SKNEPS represents the enlargement of the plane of the fluid surface used to define the search box. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + SKNEPS)).



7. DSKNEPS represents a secondary enlargement of the plane of the fluid surface used to define the search box if SKNEPS fails to find ANY structural elements. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + DSKNEPS)).
8. INTOL represents a normal direction into the fluid for the case when the fluid protrudes past the structural interface. It is defined as  $L \times INTOL$  where  $L$  is the smallest edge of the fluid element surface.
9. The BW method is summarized in the following table:

**Table of Fluid/Structure Search Control and Override METHOD=BW**

|       |  |  |
|-------|--|--|
| INFOR | Indicates the types of FSET and SSET entered                                   |  |
|       | ELEMENTS   | Element sets are used to select element fluid and structure faces. |
|       | GRIDS  | Grid sets are used to select element fluid and structure faces.    |
| FSET  | Fluid Set ID to delete or add fluid faces to fluid/structure interface         |  |
|       | ALLSET=NO  | Delete fluid element faces   |
|       | ALLSET=YES   | Add fluid element faces  |
| SSET  | Structure Set ID to delete or add structure faces to fluid/structure interface |  |

Table of Fluid/Structure Search Control and Override METHOD=BW

|          |            |   |
|----------|------------|---|
|          | ALLSET=NO  | Delete structural element faces   |
|          | ALLSET=YES | Add structural element faces  |
|          |            | A SET specification will for the specified set given override any search parameters.<br><br>Caution: When deleting items from a set, especially structural element sets an element ID may occur more than once and ALL must be removed.   |
| SRCHUNIT | REL        | Units are in the units of smallest fluid edge of current fluid face being searched.<br><br>L = smallest fluid surface edge of element being searched.<br><br>D = Diagonal from center of fluid surface to each grid<br><br>NORMAL = Value_input * L, Height of box +normal to plane of fluid.<br><br>SKNEPS = D * (1.0 + Value_input), Search plane diagonals.<br><br>DSKNEPS = D * (1.0 + Value_input) Secondary search plane diagonals<br><br>DSKNEPS is only used when SKNEPS < DSKNEPS<br><br>INTOL = Value_input * L, Height of box -normal to plane of fluid. |
|          | ABS        | Normal units are in the UNITS of the STRUCTURE. Use when distance between the structure interface and the fluid interface is same constant everywhere.<br><br>NORMAL = Value_input<br><br>SKNEPS = As defined above but projected using ABS normal units<br><br>DSKNEPS = As defined above but projected using ABS normal units<br><br>INTOL = Value_input  |
| ALLSSET  | NO         | Designed to take a modified PARAM_SKINOUT PUNCH set and DELETE elements from either the FLUID element set or the STRUCTURE element set or BOTH. Note if the FLUID set has some deleted elements and the STRUCTURE set has added elements this option may result in some change of structure elements in the set and not keep the structural set UNMODIFIED or vice versa. This is because not all of the UNMODIFIED structure elements of the set project with the reduced set of Fluid elements or vice versa.<br><br>Recommended for DELETE only.                 |
|          | YES        | Designed to take a modified PARAM_SKINOUT PUNCH set and ADD elements to either the FLUID element set or the STRUCTURE element set or BOTH. If some previously found FLUID and STRUCTURE are also deleted the algorithm keeps all the added set elements but will of course recomputed the interface for the deleted boundary so that a sensible set intersection is computed.<br><br>Recommended for DELETE ADD.  |

Table of Fluid/Structure Search Control and Override METHOD=BW

|                    |  |
|--------------------|--|
|                    | <p>Do NOT use ALLSET =YES when no SETs are specified or one SET is specified and the search algorithm is being used for the other unspecified SET as ALLSET means USE ALL and all of the elements of the unspecified SET may be used.</p>  |
| Coupling Algorithm | <ol style="list-style-type: none"> <li>1. Use the search box algorithm to locate the fluid free faces and the corresponding structural element faces.</li> <li>2. For a fluid free face and its list of structural element faces (that were determined by boxing normal to the fluid element) do as follows:             <ol style="list-style-type: none"> <li>a. For each fluid free face establish a face coordinate system.</li> <li>b. Determine the resultant pressure force for each grid on the fluid element.</li> <li>c. Resolve this resultant pressure force for a unit grid pressure to the grids of the fluid element. (Determined by principal virtual work).</li> <li>d. Using the origin of the free fluid face, determine the center of pressure.</li> <li>e. Using rigid body motion consider only a unit motion normal to the fluid face with the appropriate moment relationships, determine the resulting load distribution at the grids of each of the structural elements. The area of each structural element projected normal to the fluid element is used as a weighting function.</li> <li>f. Loop over each grid of the fluid element and accumulate the forces at the structural grids.</li> <li>g. The algorithm always maintains of rigid body equilibrium.</li> </ol> </li> <li>3. Repeat for the next fluid element and its associated group of structural elements. Accumulate the forces at the structural grids.</li> </ol> |
| Guidelines         | <ol style="list-style-type: none"> <li>1. Run default ACMODL with PARAM , SKINOUT, PUNCH</li> <li>2. View SETS both FLUID and STRUCTURE.</li> <li>3. Adjust ACMODL search parameters until close to desired interface is obtained.</li> <li>4. Modify the FSET, or SSET, or both to include and exclude Girds or Elements as seems appropriate to get final coupling elements.</li> </ol>  |

Table of Fluid/Structure Search Control and Override METHOD=BW

|  |   |
|--|---|
|  | <p>5. Run acoustic analysis with ACMODL using final FSET, SSET sets with ALLSET=YES if sets contain both deleted and added elements. Or ALLSET=NO (default) if sets only delete.</p> <p>6. The ABS option IS NOT RECOMMENDED for complex acoustic models such as body in white models with multi contouring surfaces.</p>   |
| Parallel Structural Element Meshes in the Search Box | When there are "parallel" over lapping structural meshes, the search algorithm will pick an element of the closest mesh to the fluid face. All elements connected to this first element are then checked to see if they are in the search box. Once an element, such as the elements of the outer parallel mesh, are found not to connect to the inner patch it is eliminated from the search. Thus outer patch elements tend to be excluded from the search unless some connectivity to the inner patch is detected. The ALLSET=YES is to force an interface between the listed structural elements and the nearest fluid faces. |

10. The default METHOD is the new "BW" searching algorithm that requires a special license.

The pre-Version 2004 method is selected with METHOD = "CP" in which:

- The search box is not used so the SKNEPS, DSKNEPS, INTOL, ALLSET, SRCHUNIT fields are ignored.
- If INFOR = 'ALL' (METH = 'CP' only), then both FSET and SSET must be specified and matching is checked at only those grid points referenced by FSET and SSET.
- FSET and SSET refer to grids.
- NORMAL = blank is the default (recommended),  $1 \leq \text{NORMAL} \leq 10$ . gets acceptable results. In this case, NORMAL represents a maximum cutoff value measured in physical units. When NORMAL = 'blank', Nastran will compute the cutoff value.

This field replaces the pre-Version 2004 FSTOL field. Different units are also used so pre-Version 2004 FSTOL values may need to be changed to obtain the same results.

11. METH = 'CP' is not recommended when complex acoustic cavities (such as body in white automotive models) exist in the model.

12. If Modules are present then this entry may only be specified in the main Bulk Data section.

## ACPEMCP

## Trim Component Interface Coupling and Constraints Definition

Defines the interface coupling conditions and constraints of a trim component.

**Format:**

| 1       | 2    | 3      | 4      | 5     | 6      | 7    | 8    | 9       | 10 |
|---------|------|--------|--------|-------|--------|------|------|---------|----|
| ACPEMCP | TID  | SGLUED | SSLIDE | SOPEN | SIMPER | OOC  | SPM  | SAIRGAP |    |
|         | SCUX | SCUY   | SCUZ   | SCRX  | SCRY   | SCRZ | SCFP |         |    |

**Example:**

|         |   |  |      |  |      |  |  |  |  |
|---------|---|--|------|--|------|--|--|--|--|
| ACPEMCP | 1 |  | 1002 |  | 1004 |  |  |  |  |
|---------|---|--|------|--|------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| TID        | Identification number of trim component. (Integer>0; Required)  |
| SGLUED     | Identification number of SET1/SET3 entry of grids belonging to the solid-phase and/or structure volume elements in the trim component, which are glued to a structure; see Remark 2. (Integer $\geq 0$ or Blank)                |
| SSLIDE     | Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in the sliding-contact to a structure; see Remark 3. (Integer0 or Blank)     |
| SOPEN      | Identification number of SET1/SET3 entry of grids belonging to the fluid-phase elements in the trim component, which has an open interface with the cavities; see Remark 4. (Integer $\geq 0$ or Blank)                         |
| SIMPER     | Identification number of SET1/SET3 entry of grids of the solid-phase and/or structure volume elements in the trim component, which has an impervious interface with the cavities; see Remark 5. (Integer $\geq 0$ or Blank)     |
| OOC        | Number of blocks for Out-Of-Core solver, see remark 8. (Integer>1, Default=1)'  |
| SPM        | Flag for selecting single precision MUMPS for computing reduced impedance matrix, see Remark 9. (Integer $>=0$ , default=0)   |
| SAIRGAP    | Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in contact with structure via air gap. See Remark 10. (Integer = 0 or Blank) |
| SCUX       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation X direction of output coordinate system.   |
| SCUY       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Y direction of output coordinate system.   |
| SCUZ       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Z direction of output coordinate system.   |

| Descriptor | Meaning  |
|------------|--|
| SCRX       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation X direction of output coordinate system. |
| SCRY       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Y direction of output coordinate system. |
| SCRZ       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Z direction of output coordinate system. |
| SCFP       | ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in fluid pressure.                                   |

**Remarks:**

1. ACPEMCP is a required entry for each trim component that is referenced by TRIMGRP. It must be put in the main Bulk Data section or BEGIN BULK. It specifies the boundary coupling conditions as well as the coupling degrees of freedom for computing the reduced boundary impedance matrices of the trim component.
2. A glued interface means that the trim component is completely constrained on to the structure, such as a car body. There is no relative movement between the trim component and the structure at the interface.
3. A sliding interface means that the trim component is laid on the surface of structure, such as a car body. There is no relative movement in the normal direction of the interface between the trim component and the structure. Relative sliding movement is allowed in the contact surface.
4. An open interface means that the free flow of fluid between the fluid-phase and cavities occurs at the interface.
5. An impervious or closed interface means that the fluid flow between the trim component and cavities is prohibited at the interface. This can happen when the porous material is covered by a thin film which stops the fluid flow or the cavity is in touch with a structure part of the trim component.
6. The SET1/SET3 entries must reside in the Bulk Data section introduced by Case Control, BEGIN BULK TRMC.
7. Under current PEM implementation, only 3D elements are allowed for trim components. Hence, SCRX, SCRY and SCRZ fields of ACPEMCP are not processed.
8. OOC is intended for large trim component. The number of OOC field defines the number of blocks to be used for the Schur complement evaluation which reduces the memory requirement. A side effect of OOC>1 is full RIM is generated, instead of lower triangular RIM, which is similar to PARAM,PEMFIRIM,1.
9. SPM is a flag for selecting single precision MUMPS in ACTRAN. With default value of 0, double precision MUMPS is utilized. Single precision MUMPS will be used with SPM field having value greater 0. Single precision MUMPS has the benefits of reduced memory requirement and better performance. However, single precision MUMPS may suffer minor degradation on accuracy.

10. An air gap interface means the trim component is assumed to be coupled to the structure through a very thin air layer. The thin air layer is taken into the analysis without being modeled (not being defined in the trim model file).

## ACSRCE

## Acoustic Source Specification

Defines acoustic source as a function of power vs. frequency.

$$\text{Source Strength} = \{A\} \cdot \left[ \frac{1}{2\pi f} \sqrt{\frac{8\pi CP(f)}{\rho}} \right] e^{i(\theta + 2\pi f\tau)}$$

$$C = \sqrt{B/\rho}$$

Format:

| 1      | 2   | 3        | 4                 | 5                   | 6     | 7   | 8 | 9 | 10 |
|--------|-----|----------|-------------------|---------------------|-------|-----|---|---|----|
| ACSRCE | SID | EXCITEID | DELAYI/<br>DELAYR | DPHASEI/<br>DPHASER | TP/RP | RHO | B |   |    |

Example:

|        |     |    |    |     |    |     |      |  |  |
|--------|-----|----|----|-----|----|-----|------|--|--|
| ACSRCE | 103 | 11 | 20 | 5.0 | 12 | 1.0 | 15.0 |  |  |
|--------|-----|----|----|-----|----|-----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. See Remarks 1. and 3. (Integer > 0)  |
| EXCITEID   | Identification number of DAREA, FBALOAD (in FRF Based Assembly or FBA process) or SLOAD entry set the defines $\{A\}$ . See Remark 6. (Integer > 0)                                  |
| DELAYI     | Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay $\tau$ . See Remarks 4. and 5. (Integer > 0 or blank)       |
| DELAYR     | Value of time delay $\tau$ that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)                             |
| DPHASEI    | Identification number of DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle $\theta$ . (See Remarks 4. and 5. (Integer > 0 or blank)) |
| DPHASER    | Value of phase angle $\theta$ (in degrees) that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)             |
| TP         | Identification number of a TABLEDi entry that defines power versus frequency, $P(f)$ . (Integer > 0)   |
| RP         | Value of power P to be used for all frequencies (Real, non-zero)   |
| RHO        | Density of the fluid. (Real > 0.0)   |
| B          | Bulk modulus of the fluid. (Real > 0.0)  |

#### Remarks:

1. Acoustic sources must be selected in the Case Control Section with DLOAD = SID.
2. For additional remarks, see the RLOAD1 entry description.

3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.
5. If any of the DELAYI/DELAYR or DPHASEI/DPHASER fields are blank or zero, the corresponding  $\tau$  or  $\theta$  will be zero.
6. If there is no LOADSET Case Control command, then EXCITEID may reference DAREA and SLOAD entries. If there is a LOADSET Case Control command, then EXCITEID may reference DAREA entries as well as SLOAD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

**ACTIVAT**      Defines Elements That Were Previously Deactivated and Should Be Reactivated for a Particular Subcase in SOL 600

This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase in SOL 600.

Format:

| 1       | 2  | 3     | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|-------|------|---|---|---|---|---|----|
| ACTIVAT | ID | IGEOM | ISET |   |   |   |   |   |    |

Example:

|         |   |   |     |  |  |  |  |  |  |
|---------|---|---|-----|--|--|--|--|--|--|
| ACTIVAT | 3 | 0 | 300 |  |  |  |  |  |  |
|---------|---|---|-----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of a matching Case Control ACTIVAT command defining the subcase to which these elements should be activated. (Integer; no Default)     |
| IGEOM      | Flag to control whether to update the geometry on the post file. (Integer; Default = 0)<br>0      Update the geometry.<br>1      Do not update the geometry. |
| ISET       | ID of a list of elements described by SET3 is ID=ISET to be activated. (Integer; no Default)   |

Remark:

1. This entry maps to Marc's ACTIVATE History definition option.

**ACTRIM****ACTRAN Trimmed Material Matrices for SOL 108/111**

Defines ACTRAN trimmed material matrices.

**Format:**

| 1      | 2    | 3     | 4     | 5    | 6    | 7 | 8 | 9 | 10 |
|--------|------|-------|-------|------|------|---|---|---|----|
| ACTRIM | NAME | UNIT1 | UNIT2 | SCLR | SCLI |   |   |   |    |

**Example(s):**

|        |       |    |    |     |     |  |  |  |  |
|--------|-------|----|----|-----|-----|--|--|--|--|
| ACTRIM | FLOOR | 31 | 32 | 2.0 | 0.5 |  |  |  |  |
|--------|-------|----|----|-----|-----|--|--|--|--|

| Field | Context  |
|-------|--|
| NAME  | Name of the ACTRAN trimmed material matrices. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic) |
| UNIT1 | Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)  |
| UNIT2 | Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)  |
| SCLR  | The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)                                     |
| SCLI  | The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)                                |

**Remarks:**

1. ACTRAN trimmed material matrices defined by this entry will be used for frequency response analysis if it is selected via the Case Control ACTRIM = NAME.
2. The following type of ASSIGN should be specified in the FMS section with the vacant unit number (see ASSIGN statements). The unit number cannot be selected doubly.  

```
ASSIGN INPUTT2='ACTRAN_trimmed.f70' UNIT=31
ASSIGN INPUTT4='ACTRAN_trimmed.op4' UNIT=32
```
3. Refer the ACTRAN manual for the details of exportation of the trimmed material matrix data for Nastran.

**ADAPT****Version Adaptivity Control**

Defines controls for p-version adaptive analysis.

**Format:**

| 1     | 2  | 3     | 4       | 5       | 6      | 7      | 8 | 9 | 10 |
|-------|--|-------|---------|---------|--------|--------|---|---|----|
| ADAPT | SID  | ADGEN | MAXITER | PSTRRID | PMINID | PMAXID |   |   |    |
|       | PART=name1, option1=value1, option2=value2, etc., PART=name2 |       |         |         |        |        |   |   |    |
|       | option1=value1, option2=value2, etc., PART=name3, etc.       |       |         |         |        |        |   |   |    |

**Example:**

|       |  |  |   |    |  |    |  |  |  |
|-------|--|--|---|----|--|----|--|--|--|
| ADAPT | 127  |  | 3 | 23 |  | 45 |  |  |  |
|       | PART=LOWSTR, ELSET=11, TYPE=UNIP, SIGTOL=22  |  |   |    |  |    |  |  |  |
|       | PART=HISTR, ELSET=111, ERREST=2, EPSTOL=.002 |  |   |    |  |    |  |  |  |

| Descriptor       | Meaning  | Type                                    | Default        |
|------------------|--|---|----------------|
| SID              | Adapt entry ID selected in Case Control by ADAPT command.  | Integer > 0                             | None           |
| ADGEN            | ID of the first PVAL entry generated in the adaptive process. See Remark 14.                           | Integer > PSTRRID,<br>PMINID,<br>PMAXID | 1000           |
| MAXITER          | Number of analyses performed before adaptive process is stopped.                                       | Integer > 0                             | 3              |
| PSTRRID          | ID of PVAL entry describing the starting p-order distribution.   | Integer > 0                             | None           |
| PMINID           | ID of PVAL entry describing the minimum p-order distribution. See Remark 10.                           | Integer > 0                             | PSTRRID        |
| PMAXID           | ID of PVAL entry describing the maximum p-order distribution. See Remark 10.                           | Integer > 0                             | PSTRRID        |
| optioni = valuei | Assigns a value to an option described later. See Remark 16.   |   |                |
| PART             | Part name of the elements defined in ELSET and controlled by TYPE, ERREST, ERRTOL, SIGTOL, and EPSTOL. | Character                               | PART = MODEL   |
| ELSET            | ID of the SET command under the SETS DEFINITION command. See Remark 7.                                 | Integer > 0                             | ELSET = 999999 |
| TYPE             | p-order adjustment. See Remark 3.  | Character or Integer > 0                | TYPE = EBEP    |

| Descriptor | Meaning   | Type             | Default         |
|------------|---|------------------|-----------------|
| ERREST     | Error estimator activation flag. See Remark 2.  | Integer $\geq 0$ | ERREST = 1      |
| ERRTOL     | Error tolerance. Required if MAXITER is not specified.  | 0.0 < Real < 1.0 | ERRTOL=0.01     |
| SIGTOL     | Stress tolerance. If the von Mises stress at the center of the element is below this value, the element will not participate in the error analysis. | Real $\geq 0.0$  | SIGTOL = 0.0    |
| EPSTOL     | Strain tolerance. If the von Mises strain at the center of the element is below this value, the element will not participate in the error analysis. | Real $\geq 0.0$  | EPSTOL = 1.0E-8 |

**Remarks:**

- Only one ADAPT entry may be specified. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
- The error estimator is activated by ERREST = 1 and is based on strain energy density sensitivity and stress discontinuity in neighboring elements. ERREST = 0 means no error estimation will be performed on the PART.
- The types of p-order adjustment are:

| Type | Description   |
|------|---|
| EBEP | The p-order will increase only in the elements that are required by the error analysis.   |
| UNIP | If any element in the group has an error larger than the tolerance, all elements will be increased by one order in each direction.  |
| NOCH | The p-order of the group does not change during the iterations.   |
| LIST | The PVAL distribution specified as PSTRUPTID is used for the first iteration. The user is required to provide PVAL entries with IDs starting with ADGEN, and these p-distributions will be used in the following iterations |

- If a PVAL ID is not specified for PSTRUPTID, PMINID, or PMAXID, then this is equivalent to no change at the last PVAL ID found for the element.
- The elements specified in the SET could overlap. In this case, the highest  $p_1$ , highest  $p_2$ , highest  $p_3$  (the polynomial order of the elements in three directions) determined by the error estimator will be used.
- n restart, PMINID and PMAXID must not refer to any PVAL identification number that was generated in the previous run(s). Also, PARAM,PVALINIT must specify the desired PVAL identification number from which to restart.
- If an element in the SET does not have a PVAL for PSTRUPTID or PMINID or PMAXID, it will be excluded from the adaptivity process.

8. SET = 999999 is a reserved set that includes all elements.
9. The user can specify as many PARTs as needed.
10. Each finite element has to have a unique PVAL for PSTRUPTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest pi field (highest  $p_2$  if same  $p_1$  and highest  $p_3$  if same  $p_1$  and  $p_2$ ) and the lowest CID value.
11. The p-distribution for an element specified by the PVAL entry referenced by PMAXID must be larger than the distribution specified by the PSTRUPTID, which must be larger than the distribution specified by the PMINID. A warning message will be issued if these conditions are not met, and the data is reset.
12. The solution vector of all the elements listed in the SET entries for all loads and boundary conditions will be used in the error estimation. New p values are generated for all the elements.
13. When ERREST = 0, no error analysis is performed. The p-value of the elements in the set are increased uniformly starting from p-values specified on the PVAL entry referenced by PSTRUPTID up to values specified on the PVAL entry referenced by PMAXID.
14. The intermediate PVAL entries generated will have an ID starting with ADGEN; thus, ADGEN must be larger than PSTRUPTID, PMINID, and PMAXID.
15. The displacement and stress output can be requested by a DATAREC Case Control command.
16. Each optioni = valuei must be specified on the same entry. In other words, optioni and valuei may not be specified on two separate continuation entries.

**ADUMi****Dummy Element Attributes**

Defines attributes of the dummy elements ( $3 \leq i \leq 9$ ).

**Format:**

| 1     | 2  | 3  | 4  | 5  | 6    | 7 | 8 | 9 | 10 |
|-------|----|----|----|----|------|---|---|---|----|
| ADUMi | NG | NC | NP | ND | ELNM |   |   |   |    |

**Example:**

|       |    |   |   |   |        |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|
| ADUM8 | 18 | 0 | 5 | 0 | CRAC2D |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NG         | Maximum number of grid points that may be connected by DUMi dummy element (Integer > 0.)   |
| NC         | Number of additional fields (Ai) on the CDUMi connection entry. (Integer $\geq 0$ )  |
| NP         | Number of additional fields (Ai) on the PDUMi property entry. ( $24 \geq$ Integer $\geq 0$ )   |
| ND         | Number of displacement components at each grid point used in generation of the differential stiffness matrix. Zero implies no differential stiffness. (Integer 3 or 6) |
| ELNM       | The name of the element connection and property entry. In the example above, the connection entry is named "CRAC2D" and the property entry is named "PRAC2D".          |

**Remarks:**

ADUM8 and ADUM9 are used exclusively for the CRAC2D and CRAC3D entries respectively.

**AECOMP****Component for an Integrated Load Monitor Point**

Defines a component for use in monitor point definition or external splines.

**Format:**

| 1      | 2       | 3        | 4       | 5       | 6       | 7       | 8       | 9       | 10 |
|--------|---------|----------|---------|---------|---------|---------|---------|---------|----|
| AECOMP | NAME    | LISTTYPE | LISTID1 | LISTID2 | LISTID3 | LISTID4 | LISTID5 | LISTID6 |    |
|        | LISTID7 | -etc.-   |         |         |         |         |         |         |    |

**Example:**

|        |      |        |      |      |  |  |  |  |  |
|--------|------|--------|------|------|--|--|--|--|--|
| AECOMP | WING | AELIST | 1001 | 1002 |  |  |  |  |  |
|--------|------|--------|------|------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| NAME       | A character string of up to eight characters identifying the component. (Character)   |
| LISTTYPE   | One of CAERO, AELIST or CMPID for aerodynamic components and SET1 for structural components. Aerodynamic components are defined on the aerodynamic ks-set mesh while the structural components are defined on the g-set mesh. See Remarks 2. and 4. |
| LISTIDi    | The identification number of either SET1, AELIST or CAEROi entries that define the set of grid points that comprise the component. See Remarks 2. and 4.  |

**Remarks:**

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. If the component is defined on the structure, LISTIDs must refer to SET1 entry(ies) that define the list of associated GRID points. For the AELIST or CAERO option, the LISTIDs must refer to AELIST or CAERO i entries, respectively. Note that, for DLM models (CAERO1/2), the set of points defined by the AELIST are the box identification numbers. For example, if the control surface's grids are desired, the same AELIST used for the AESURF can be referred to here. An AECOMP component must be defined as either an aerodynamic mesh component or a structural component. The two mesh classes cannot be combined into a single component.
3. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.
4. If LISTTYPE = CMPID, LISTID1 identifies the associated set of AEQUAD4, AETRIA3 elements that define the aero component.

**AECOMPL****Component for an Integrated Load Monitor Point**

Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| AECOMPL | NAME   | LABEL1 | LABEL2 | LABEL3 | LABEL4 | LABEL5 | LABEL6 | LABEL7 |    |
|         | LABEL8 | -etc.- |        |        |        |        |        |        |    |

**Example:**

|         |       |      |      |         |  |  |  |  |  |
|---------|-------|------|------|---------|--|--|--|--|--|
| AECOMPL | HORIZ | STAB | ELEV | BALANCE |  |  |  |  |  |
|---------|-------|------|------|---------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NAME       | A character string of up to eight characters Identifying the component. (Character)                                    |
| LABELi     | A string of 8 characters referring to the names of other components defined by either AECOMP or other AECOMPL entries. |

**Remarks:**

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

**AEDW****Parametric Normal Wash Loading for Aerodynamics**

Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

**Format:**

| 1    | 2    | 3     | 4     | 5    | 6    | 7     | 8 | 9 | 10 |
|------|------|-------|-------|------|------|-------|---|---|----|
| AEDW | MACH | SYMXZ | SYMXY | UXID | DMIJ | DMIJI |   |   |    |

**Example:**

|      |      |      |       |     |      |  |  |  |  |
|------|------|------|-------|-----|------|--|--|--|--|
| AEDW | 0.90 | SYMM | ASYMM | 101 | ALP1 |  |  |  |  |
|------|------|------|-------|-----|------|--|--|--|--|

| Descriptor  | Meaning  |
|-------------|--|
| MACH        | The Mach number for this force, see Remark 2. (Real $\geq 0.0, \neq 1.0$ )   |
| SYMXZ,SYMXY | The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).   |
| UXID        | The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector. |
| DMIJ        | The name of a DMI or DMIJ entry that defines the downwash.   |
| DMIJI       | The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.                                       |

**Remarks:**

1. The AEDW, AEFORCE and AEPRESS are associated with the current AECOMFIG using either Case Control (if in the main Bulk Data Section) or using the BEGIN AECOMFIG=<config> if in a partition of the Bulk Data.
2. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.
3. Mach numbers  $> 1.0$  require that the supersonic aerodynamic option be available.

**AEFACT****Aerodynamic Lists**

Defines real numbers for aeroelastic analysis.

**Format:**

| 1      | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|--------|----|----|----|----|----|----|
| AEFACT | SID | D1 | D2     | D3 | D4 | D5 | D6 | D7 |    |
|        | D8  | D9 | -etc.- |    |    |    |    |    |    |

**Example:**

|        |    |    |    |     |  |  |  |  |  |
|--------|----|----|----|-----|--|--|--|--|--|
| AEFACT | 97 | .3 | .7 | 1.0 |  |  |  |  |  |
|--------|----|----|----|-----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Unique Integer > 0) |
| Di         | Number. (Real)                                  |

**Remarks:**

1. AEFACt entries must be selected by a CAEROi, PAEROi or SPLINEX entry.
2. Embedded blank fields are not allowed.
3. To specify division points, there must be one more division point than the number of divisions.
4. When referenced by the CAERO3 entry, AEFACt defines the aerodynamic grid points. The ID number of the first point defined by each AEFACt entry is the value of the CAERO3 ID that selected the AEFACt entry. The ID of each following point defined on the AEFACt is incremented by 1.

**AEFORCE**

## Parametric Force for Aerodynamics

Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector. This force vector may be defined on either the aerodynamic mesh (ks-set) or the structural mesh (g-set). The force vector will be used in static aeroelastic trim.

**Format:**

| 1       | 2    | 3     | 4     | 5    | 6    | 7    | 8    | 9    | 10 |
|---------|------|-------|-------|------|------|------|------|------|----|
| AEFORCE | MACH | SYMXZ | SYMXY | UXID | MESH | LSET | DMIK | PERQ |    |

**Example:**

|         |      |      |       |     |      |  |      |  |  |
|---------|------|------|-------|-----|------|--|------|--|--|
| AEFORCE | 0.90 | SYMM | ASYMM | 101 | AERO |  | BETA |  |  |
|---------|------|------|-------|-----|------|--|------|--|--|

| Descriptor  | Meaning  |
|-------------|--|
| MACH        | The Mach number for this force. (Real $\geq 0.0$ , $\neq 1.0$ )  |
| SYMXZ,SYMXY | The symmetry conditions for this force vector. One of SYMM, ASYMM or ANTI. (Character)   |
| UXID        | The identification number of a UXVEC entry that defines the control parameter vector associated with this force vector.                        |
| MESH        | One of AERO or STRUCT that declares whether the force vector is defined on the aerodynamic ks-set mesh or the structural g-set mesh.           |
| LSET        | SID of a load set that defines the vector. See Remark 2. (Integer $> 0$ if MESH=STRUCT)  |
| DMIK        | The name of a DMIK entry that defines the aerodynamic force vector. See Remark 3. (Character; Required if MESH=AERO)                           |
| PERQ        | The string PERQ or blank. If PERQ and MESH=STRUCT, the input FORCE set is multiplied by the dynamic pressure. (Default = blank; see Remark 4.) |

**Remarks:**

1. The AEFORCE is associated with the current AECONFIG and must be entered for the appropriate Mach numbers and aerodynamic symmetries.
2. For the STRUCT mesh option, the LSET can refer to any existing load type (e.g., FORCE1, PLOAD4 or LOAD) that is available to define static loads.
3. For the AERO mesh option, the DMIK Bulk Data are used. Any forces associated with the aerodynamic model’s permanently SPC’d degrees-of-freedom (which are dependent on the type of aerodynamic model being used) will be ignored.
4. For the AERO mesh option, the PERQ field is not used. The FORCE data input on the DMIK entry is always multiplied by dynamic pressure.

**AEGRID****Aerodynamic Grid Point**

Defines the location of an aerodynamic grid point.

**Format:**

| 1      | 2   | 3  | 4  | 5  | 6  | 7  | 8    | 9 | 10 |
|--------|-----|----|----|----|----|----|------|---|----|
| AEGRID | GID | CP | X1 | X2 | X3 | CD | ASID |   |    |

**Example:**

|        |   |   |     |     |     |   |   |  |  |
|--------|---|---|-----|-----|-----|---|---|--|--|
| AEGRID | 1 | 1 | 2.1 | 3.2 | 2.0 | 3 | 4 |  |  |
|--------|---|---|-----|-----|-----|---|---|--|--|

| Descriptor | Meaning   |
|------------|---|
| GID        | Grid Point Identification Number ( $0 < \text{Integer} < 100,000,000$ )   |
| CP         | Identification number of a coordinate system in which the location of the grid point is defined. ( $\text{Integer} \geq 0$ or blank)          |
| X1, X2, X3 | Location of the grid point in the coordinate system CP.   |
| CD         | Identification number of coordinate system in which the degrees-of-freedom of the grid point are defined. ( $\text{Integer} \geq 0$ or blank) |
| ASID       | Identification number of an AESCALE Bulk Data entry. ( $\text{Integer} > 0$ or blank)   |

**Remarks:**

1. All grid point identification numbers must be unique with respect to all other aerodynamic grid point identification numbers.
2. If both CP and ASID are defined, coordinates are first scaled and subsequently transformed to the basic coordinate system.
3. The meaning of X1, X2 and X3 depends on the type of coordinate system CP. (See Remark 2 of the GRID entry).
4. A zero or blank in the CP and CD fields refers to the basic coordinate system.
5. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

**AELINK****Links Aeroelastic Variables**

Defines relationships between or among AESTAT and AESURF entries, such that:

$$u^D + \sum_{i=1}^n C_i u_i^I = 0.0$$

Where:

$u^D$  = dependent variable

$u_i^I$  = independent variable

Format:

| 1      | 2     | 3     | 4      | 5  | 6     | 7  | 8     | 9  | 10 |
|--------|-------|-------|--------|----|-------|----|-------|----|----|
| AELINK | ID    | LABLD | LABL1  | C1 | LABL2 | C2 | LABL3 | C3 |    |
|        | LABL4 | C4    | -etc.- |    |       |    |       |    |    |

Example:

|        |    |       |       |      |  |  |  |  |  |
|--------|----|-------|-------|------|--|--|--|--|--|
| AELINK | 10 | INBDA | OTBDA | -2.0 |  |  |  |  |  |
|--------|----|-------|-------|------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | If an integer > 0 is specified, this is the TRIM set ID selected in Case Control and the AELINK only applies to that subcase. If an integer value of 0 or the character string "ALWAYS" is specified, this AELINK is applicable to all subcases. (Integer $\geq 0$ or the "ALWAYS" character string.) |
| LABLD      | Character string to identify the dependent aerodynamic variable. (Character)  |
| LABLi      | Character string to identify the i-th independent aerodynamic variable. (Character)   |
| Ci         | Linking coefficient for the i-th variable. (Real)   |

Remarks:

1. If the ID is a positive integer, the AELINK entry (or entries) is selected by the TRIM-ID in Case Control.
2. If the ID is 0 or the character string ALWAYS, the linking relationship applies to all subcases.
3. The entry constrains the dependent variable to be a linear combination of the independent variables.
4. LABLD data must be unique for a given ID or if ID=0 or AWAYS is being used (i.e., the variable cannot be constrained more than once).
5. LABLD and LABLi refer to AEPARM, AESTAT or AESURF Bulk Data entries.

**AELIST****Aerodynamic List**

Defines a list of aerodynamic elements or grid ID's.

**Format:**

| 1      | 2   | 3      | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|--------|----|----|----|----|----|----|----|
| AELIST | SID | E1     | E2 | E3 | E4 | E5 | E6 | E7 |    |
|        | E8  | -etc.- |    |    |    |    |    |    |    |

**Example:**

|        |      |      |      |      |      |      |      |      |  |
|--------|------|------|------|------|------|------|------|------|--|
| AELIST | 75   | 1001 | THRU | 1075 | 1101 | THRU | 1109 | 1201 |  |
|        | 1202 |      |      |      |      |      |      |      |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)                                    |
| Ei         | List of aerodynamic box ID's or aerodynamic grid ID's (Integer>0 or "THRU") |

**Remarks:**

1. The AELIST entry can be referenced by AESURF, AECOMP, SPLINEi, SPLINRB, SPBLND2 and SPRELAX entries. Refer to these entries for the meaning of the data provided.
2. When the "THRU" option is used, all intermediate grid points must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
3. Intervening blank fields are not allowed.

**AELISTC**

## Character Item List

Defines a list of 8-character strings.

Format:

| 1       | 2   | 3      | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|-----|--------|----|----|----|----|----|----|----|
| AELISTC | SID | C1     | C2 | C3 | C4 | C5 | C6 | C7 |    |
|         | C8  | -etc.- |    |    |    |    |    |    |    |

Example:

|         |     |     |          |  |  |  |  |  |  |
|---------|-----|-----|----------|--|--|--|--|--|--|
| AELISTC | 101 | FBS | STRING12 |  |  |  |  |  |  |
|---------|-----|-----|----------|--|--|--|--|--|--|

**Descriptor      Meaning**

SID              Set identification number. (Integer > 0)

Ci              List of 8-character strings.

Remark:

1. Intervening blank fields are not allowed.

**AEPARM****General Controller for Use in Trim**

Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point). The forces associated with this controller will be derived from AEDW, AEFORCE and AEPRESS input data.

Format:

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| AEPARM | ID | LABEL | UNITS |   |   |   |   |   |    |

Example:

|        |   |        |     |  |  |  |  |  |  |
|--------|---|--------|-----|--|--|--|--|--|--|
| AEPARM | 5 | THRUST | LBS |  |  |  |  |  |  |
|--------|---|--------|-----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Controller identification number. (Integer > 0)                        |
| LABEL      | Controller name. See Remark 1. (Character)                             |
| UNITS      | Label used to describe the units of the controller values. (Character) |

Remarks:

1. Controller LABELs that comprise the unique set relative to all the AESURF, AESTAT and AEPARM entries will define the set of trim variable degrees-of-freedom for the aeroelastic model.
2. Unit labels are optional and are only used to label outputs. No units will be associated with the controller if left blank.

**AEPRESS**

## Parametric Pressure Loading for Aerodynamics

Defines a vector of pressure/unit dynamic pressure associated with a particular control vector. From this pressure vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

**Format:**

| 1       | 2    | 3     | 4     | 5    | 6    | 7     | 8 | 9 | 10 |
|---------|------|-------|-------|------|------|-------|---|---|----|
| AEPRESS | MACH | SYMXZ | SYMXY | UXID | DMIJ | DMIJI |   |   |    |

**Example:**

|         |      |      |       |     |      |  |  |  |  |
|---------|------|------|-------|-----|------|--|--|--|--|
| AEPRESS | 0.90 | SYMM | ASYMM | 101 | ALP1 |  |  |  |  |
|---------|------|------|-------|-----|------|--|--|--|--|

| Descriptor  | Meaning  |
|-------------|--|
| MACH        | The Mach number for this force, see Remark 2. (Real $\geq 0.0$ , $\neq 1.0$ )  |
| SYMXZ,SYMXY | The symmetry of this force vector. One of SYMM, ASYMM or ANTI. (Character)   |
| UXID        | The identification number of a UXVEC entry that defines the control parameter vector associated with this pressure vector. |
| DMIJ        | The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure.                                       |
| DMIJI       | The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.                                       |

**Remarks:**

1. The AEDW, AEFORCE, and AEPRESS are associated with the current AECONFIG using Case Control.
2. Mach numbers  $> 1.0$  require that the supersonic aerodynamic option be available.
3. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.

**AEQUAD4****Quadrilateral Aerodynamic Element Connection**

Defines the connectivity of a quadrilateral aerodynamic element.

**Format:**

| 1       | 2   | 3     | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|-----|-------|----|----|----|----|---|---|----|
| AEQUAD4 | EID | CMPID | G1 | G2 | G3 | G4 |   |   |    |

**Example:**

|         |     |   |    |    |    |    |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|
| AEQUAD4 | 100 | 2 | 12 | 32 | 41 | 50 |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element Identification Number. (0 < Integer < 100,000,000)            |
| CMPID      | Aerodynamic Component Identification Number. (Integer > 0)            |
| Gi         | Grid Point Identification Numbers of Connection Points. (Integer > 0) |

**Remarks:**

1. The geometry of a quadrilateral aerodynamic element may collapse to a triangle, i.e., two connection points may have the same geometric location. However, all four grid point identification numbers must be different.
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

**AERO****Aerodynamic Physical Data**

Gives basic aerodynamic parameters for unsteady aerodynamics.

Format:

| 1    | 2     | 3        | 4    | 5      | 6     | 7     | 8 | 9 | 10 |
|------|-------|----------|------|--------|-------|-------|---|---|----|
| AERO | ACSID | VELOCITY | REFC | RHOREF | SYMXZ | SYMXY |   |   |    |

Example:

|      |   |       |      |      |   |    |  |  |  |
|------|---|-------|------|------|---|----|--|--|--|
| AERO | 3 | 1.3+4 | 100. | 1.-5 | 1 | -1 |  |  |  |
|------|---|-------|------|------|---|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ACSID      | Aerodynamic coordinate system identification. See Remark 2. (Integer $\geq 0$ ; Default is the basic coordinate system)  |
| VELOCITY   | Velocity for aerodynamic force data recovery and to calculate the BOV parameter. See Remark 5. (Real $> 0.0$ or blank)   |
| REFC       | Reference length for reduced frequency. (Real $> 0.0$ )  |
| RHOREF     | Reference density. (Real $> 0.0$ )   |
| SYMXZ      | Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)                             |
| SYMXY      | The symmetry key for the aero coordinate x-y plane can be used to simulate ground effect. (Integer = -1 for symmetry, 0 for no symmetry, and +1 for antisymmetry; Default = 0) |

Remarks:

1. This entry is required for aerodynamic problems. Only one AERO entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction.
3. Set SYMXY = -1 to simulate ground effect.
4. PARAM,WTMASS does not affect aerodynamic matrices. RHOREF must be input in mass units.
5. VELOCITY is used only in aeroelastic response analysis, and it must be equal to V on the GUST Bulk Data entry.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

**AEROS****Static Aeroelasticity Physical Data**

Defines basic parameters for static aeroelasticity.

**Format:**

| 1     | 2     | 3     | 4    | 5    | 6    | 7     | 8     | 9 | 10 |
|-------|-------|-------|------|------|------|-------|-------|---|----|
| AEROS | ACSID | RCSID | REFC | REFB | REFS | SYMXZ | SYMXY |   |    |

**Example:**

|       |    |    |     |      |       |   |  |  |  |
|-------|----|----|-----|------|-------|---|--|--|--|
| AEROS | 10 | 20 | 10. | 100. | 1000. | 1 |  |  |  |
|-------|----|----|-----|------|-------|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ACSID      | Aerodynamic coordinate system identification. See Remark 2. (Integer $\geq 0$ ; Default is the basic coordinate system)   |
| RCSID      | Reference coordinate system identification for rigid body motions. (Integer $\geq 0$ ; Default is the basic coordinate system)  |
| REFC       | Reference chord length. (Real $> 0.0$ )   |
| REFB       | Reference span. (Real $> 0.0$ )   |
| REFS       | Reference wing area. (Real $> 0.0$ )  |
| SYMXZ      | Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)                              |
| SYMXY      | The symmetry key for the aero coordinate x-y plane can be used to simulate ground effects. (Integer = +1 for antisymmetry, 0 for no symmetry, and -1 for symmetry; Default = 0) |

**Remarks:**

1. This entry is required for static aeroelasticity problems. Only one AEROS entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction (T1).
3. The RCSID must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system.
4. REFB should be full span, even on half-span models.
5. REFS should be half area on half-span models.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

**AESCALE****Aerodynamic Grid Point Scaling**

Defines reference lengths to scale aerodynamic grid points.

**Format:**

| 1       | 2    | 3     | 4     | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|------|-------|-------|-------|---|---|---|---|----|
| AESCALE | ASID | X1REF | X2REF | X3REF |   |   |   |   |    |

**Example:**

|         |     |   |   |   |  |  |  |  |  |
|---------|-----|---|---|---|--|--|--|--|--|
| AESCALE | 100 | 1 | 1 | 1 |  |  |  |  |  |
|---------|-----|---|---|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ASID       | Identification Number called out on an AEGRID entry. (0 < Integer < 100,000,000) |
| X1REF      | Reference length to scale X1. (Real or blank; Default = 1.0)                     |
| X2REF      | Reference length to scale X2. (Real or blank; Default = 1.0)                     |
| X3REF      | Reference length to scale X3. (Real or blank; Default = 1.0)                     |

**Remark:**

1. The scaled coordinates are computed from  $XiS = Xi \cdot XiREF$ .
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

**AESTAT****Static Aeroelasticity Trim Variables**

Specifies rigid body motions to be used as trim variables in static aeroelasticity.

**Format:**

| 1      | 2  | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|---|---|---|---|---|---|----|
| AESTAT | ID | LABEL |   |   |   |   |   |   |    |

**Example:**

|        |      |        |  |  |  |  |  |  |  |
|--------|------|--------|--|--|--|--|--|--|--|
| AESTAT | 5001 | ANGLEA |  |  |  |  |  |  |  |
|--------|------|--------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of an aerodynamic trim variable degree-of-freedom. See Remark 1. (Integer > 0)               |
| LABEL      | An alphanumeric string of up to eight characters used to identify the degree-of-freedom. See Remark 1. (Character) |

**Remarks:**

1. The degrees-of-freedom defined with this entry represent rigid body motion in the reference coordinate system defined on the AEROS entry. The standard labels that define the various rigid body motions are as follows:

**Table 1      Standard Labels Defining Rigid Body Motions**

| LABEL  | Degree-of-Freedom Motion | Description                  |
|--------|--------------------------|------------------------------|
| ANGLEA | $u_r$ (R2)               | Angle of Attack              |
| SIDES  | $u_r$ (R3)               | Angle of Sideslip            |
| ROLL   | $\dot{u}_r$ (R1)         | Roll Rate = $p_b/2V$         |
| PITCH  | $\dot{u}_r$ (R2)         | Pitch Rate = $q_c/2V$        |
| YAW    | $\dot{u}_r$ (R3)         | Yaw Rate = $r_b/2V$          |
| URDD1  | $\ddot{u}_r$ (T1)        | Longitudinal (See Remark 3.) |
| URDD2  | $\ddot{u}_r$ (T2)        | Lateral                      |
| URDD3  | $\ddot{u}_r$ (T3)        | Vertical                     |
| URDD4  | $\ddot{u}_r$ (R1)        | Roll                         |
| URDD5  | $\ddot{u}_r$ (R2)        | Pitch                        |
| URDD6  | $\ddot{u}_r$ (R3)        | Yaw                          |

These reserved names may be defined on the AEPARM entry instead, in which case the incremental load due to the unit perturbation of the rigid body degree-of-freedom (as it will with AESTAT). See the AEPARM, AEPRESS, and AEFORCE entries.

2. The degrees-of-freedom defined with this entry are variables in the static aeroelastic trim solution, unless they are constrained by referencing them with a TRIM Bulk Data entry.
3. If a label other than those above is specified, then the user must either generate the corresponding forces with an AELINK or via a DMI Bulk Data entry along with a DMAP alter that includes the DMIIN module and additional statements to merge into the appropriate matrices. Or, using AEPARM and the AEDW, AEPRESS, and/or AEFORCE, you can accomplish this purpose without the need for any alters.

**AESURF****Aerodynamic Control Surface**

Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points. The forces associated with this controller will be derived from rigid rotation of the aerodynamic model about the hinge line(s) and from AEDW, AEFORCE and AEPRESS input data. The mass properties of the control surface can be specified using an AESURFS entry.

**Format:**

| 1      | 2     | 3     | 4     | 5     | 6      | 7      | 8      | 9      | 10 |
|--------|-------|-------|-------|-------|--------|--------|--------|--------|----|
| AESURF | ID    | LABEL | CID1  | ALID1 | CID2   | ALID2  | EFF    | LDW    |    |
|        | CREFC | CREFS | PLLIM | PULIM | HMLLIM | HMULIM | TQLLIM | TQULIM |    |

**Example:**

|        |      |       |     |     |        |       |    |    |  |
|--------|------|-------|-----|-----|--------|-------|----|----|--|
| AESURF | 6001 | ELEV  | 100 | 100 | 200    | 200   |    |    |  |
|        | 10.0 | 180.0 |     |     | -1.4E4 | 1.2E4 | 20 | 30 |  |

| Descriptor       | Meaning   |
|------------------|---|
| ID               | Controller identification number. (Integer > 0)   |
| LABEL            | Controller name. (Character)  |
| CIDi             | Identification number of a rectangular coordinate system with a y-axis that defines the hinge line of the control surface component. (Integer > 0)  |
| ALIDi            | Identification of an AELIST Bulk Data entry that identifies all aerodynamic elements that make up the control surface component. (Integer > 0)  |
| EFF              | Control surface effectiveness. See Remark 4. (Real ≠ 0.0; Default = 1.0)  |
| LDW              | Linear downwash flag. See Remark 2. (Character, one of LDW or NOLDW; Default = LDW).  |
| CREFC            | Reference chord length for the control surface. (Real > 0.0; Default = 1.0)   |
| CREFS            | Reference surface area for the control surface. (Real > 0.0; Default = 1.0)   |
| PLLLIM,PULIM     | Lower and upper deflection limits for the control surface in radians. (Real, Default = ± π/2)   |
| HMULLIM, HMULI M | Lower and upper hinge moment limits for the control surface in force-length units. (Real; Default = no limit)   |
| TQLLIM,TQULIM    | Set identification numbers of TABLEDi entries that provide the lower and upper deflection limits for the control surface as a function of the dynamic pressure. (Integer > 0; Default = no limit) |

**Remarks:**

1. The ID on AESURF, AESTAT, and AEPARM entries are ignored. AESURFS can be used to define mass properties of the control surface.
2. The degrees-of-freedom defined on this entry represent a rigid body rotation of the control surface components about their hinge lines. In the default LDW (Linear DownWash) case, the downwash due to a unit perturbation of the control surface will be computed as part of the database. In the NOLDW case, the user must prescribe the controller's effects by direct definition of the induced forces using the AEPRESS, AEDW and/or AEFORCE entries.
3. Either one or two control surface components may be defined.
4. If EFF is specified, then the forces produced by this surface are modified by EFF (e.g., to achieve a 40% reduction, specify EFF=0.60).
5. The continuation is not required.
6. The CREFC and CREFS values are only used in computing the nondimensional hinge moment coefficients.
7. Position limits may be specified using either PiLIM or TQiLIM, but not both.
8. Position and hinge moment limits are not required.

**AESURFS**

## Structural Grids on an Aerodynamic Control Surface

Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry. The mass associated with these structural nodes define the control surface moment(s) of inertia about the hinge line(s).

**Format:**

| 1       | 2  | 3     | 4 | 5     | 6 | 7     | 8 | 9 | 10 |
|---------|----|-------|---|-------|---|-------|---|---|----|
| AESURFS | ID | LABEL |   | LIST1 |   | LIST2 |   |   |    |

**Example:**

|         |      |      |  |      |  |      |  |  |  |
|---------|------|------|--|------|--|------|--|--|--|
| AESURFS | 6001 | ELEV |  | 6002 |  | 6003 |  |  |  |
|---------|------|------|--|------|--|------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Controller identification number, see Remark 1. (Integer > 0)  |
| LABEL      | Controller name, see Remark 1. (Character)   |
| LISTi      | Identification number of a SET1 entry that lists the structural grid points that are associated with this component of this control surface. (Integer > 0) |

**Remarks:**

1. The LABEL on the AESURFS entry must match one on an AESURF entry. The ID is ignored.
2. The mass of the GRID points listed on the SETi entries is used to compute the mass moment of inertia of the control surface about its i-th hinge line. The presence of these data will allow the hinge moments to include the inertial forces in the computations. These data are optional, and, if omitted, result in hinge moments which include only the applied, aeroelastically corrected, forces.
3. These data will be associated to a structural superelement by grid list or partitioned SUPER=<seid> if the AESURFS is defined in the main bulk data section.

**AETRIA3****Triangular Aerodynamic Element Connection**

Defines the connectivity of a triangular aerodynamic element.

**Format:**

| 1       | 2   | 3     | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|---------|-----|-------|----|----|----|---|---|---|----|
| AETRIA3 | EID | CMPID | G1 | G2 | G3 |   |   |   |    |

**Example:**

|         |      |   |     |     |     |  |  |  |  |
|---------|------|---|-----|-----|-----|--|--|--|--|
| AETRIA3 | 3768 | 8 | 368 | 872 | 999 |  |  |  |  |
|---------|------|---|-----|-----|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element Identification Number. (0 < Integer < 100,000,000)            |
| CMPID      | Aerodynamic Component Identification Number. (Integer > 0)            |
| Gi         | Grid Point Identification Numbers of Connection Points. (Integer > 0) |

**Remarks:**

1. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

**ALIASM**

## Alias Element Formulation in SOL 600

Allows selected elements which normally use a default formulation to be aliased to a different formulation in SOL 600.

Format:

| 1      | 2    | 3   | 4    | 5   | 6  | 7  | 8 | 9 | 10 |
|--------|------|-----|------|-----|----|----|---|---|----|
| ALIASM | TYPE | ID1 | THRU | ID2 | BY | ID |   |   |    |

Example:

|        |     |     |      |     |    |   |  |  |  |
|--------|-----|-----|------|-----|----|---|--|--|--|
| ALIASM | 139 | 101 | THRU | 300 | BY | 2 |  |  |  |
|        | 138 | 501 | THRU | 600 |    |   |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ITYPE      | Desired element formulation type. (Integer; no Default.) See Marc Vol B for a list of element types in the example, type 139 is a bilinear 3-node thin shell. ITYPE = -1 means to use the companion reduced integration element formulation (if it exists) default full integration formulation. (See Note 4) |
| ID1        | Starting element number to be aliased. (Integer; no Default)  |
| THRU       | Enter "THRU" if a range of elements is to be specified. (Character; no Default)   |
| ID2        | Ending element number to be aliased. (Integer; no Default)  |
| BY         | Enter "BY" if the range of elements is not to be incremented by one.  |
| ID3        | Element "increment by" value. (Integer; Default = 1; must be positive)  |

Remarks:

1. This entry should only be used if the Marc GEOMETRY entries are identical for the original and new element types.
2. This entry may be repeated as often as desired to identify all elements requiring aliases.
3. ITYPE and ID1 are required fields. All other data fields may be blank.
4. This entry may not be combined with parameters MRALIAS, MALIAS02, MALIAS03, etc.
5. If all elements of a particular type should have alias values, it may be easier to use parameters MRALIAS, MALIAS02, MALIAS03, etc. than this entry.
6. For ITYPE = -1, the following reduced integration element types will be used:

| Original Type | Reduced Integration Type | Description                            |
|---------------|--------------------------|--|
| 26            | 53                       | Plane stress 8-node quad               |
| 27            | 54                       | Plane strain 8-node quad               |
| 28            | 55                       | Axisymmetric 8-node quad               |
| 29            | 56                       | Generalized plane strain 8+2 node quad |
| 21            | 57                       | 20-node brick                          |
| 32            | 58                       | Plane strain Herrmann quad             |
| 33            | 59                       | Axisymmetric Herrmann 8-node quad      |
| 34            | 60                       | Plane strain Herrmann 8+2 node quad    |
| 35            | 61                       | Herrmann 20-node brick                 |
| 62            | 73                       | Axisymmetric 8-node quad, arbitrary    |
| 63            | 74                       | Axisymmetric Herrman 8-node quad       |
| 3             | 114                      | Plane stress quad                      |
| 11            | 115                      | Plane strain quad                      |
| 10, 20        | 116                      | Axisymmetric ring (quad)               |
| 7             | 117                      | 8-node brick                           |
| 80            | 118                      | Plane strain incompressible quad       |
| 82, 83        | 119                      | Axisymmetric incompressible ring       |
| 84            | 120                      | 8-node incompressible brick            |
| 75            | 140                      | 4-node quad                            |

**ASET****Degrees-of-Freedom for the a-set**

Defines degrees-of-freedom in the analysis set (a-set).

**Format:**

| 1    | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|------|-----|----|-----|----|-----|----|-----|----|----|
| ASET | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|      |    |   |    |      |   |   |  |  |  |
|------|----|---|----|------|---|---|--|--|--|
| ASET | 16 | 2 | 23 | 3516 | 1 | 4 |  |  |  |
|------|----|---|----|------|---|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| IDi        | Grid or scalar point identification number. (Integer > 0)   |
| Ci         | Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |

**Remarks:**

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive a-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

**ASET1**

## Degrees-of-Freedom For the a-set, Alternate Form of ASET Entry

Defines degrees-of-freedom in the analysis set (a-set).

Format:

| 1     | 2   | 3   | 4    | 5      | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|------|--------|-----|-----|-----|-----|----|
| ASET1 | C   | ID1 | ID2  | ID3    | ID4 | ID5 | ID6 | ID7 |    |
|       | ID8 | ID9 | ID10 | -etc.- |     |     |     |     |    |

Example:

|       |     |   |   |   |    |   |   |   |  |
|-------|-----|---|---|---|----|---|---|---|--|
| ASET1 | 345 | 2 | 1 | 3 | 10 | 9 | 6 | 5 |  |
|       | 7   | 8 |   |   |    |   |   |   |  |

Alternate Format and Example:

|       |        |     |        |     |  |  |  |  |  |
|-------|--------|-----|--------|-----|--|--|--|--|--|
| ASET1 | C      | ID1 | "THRU" | ID2 |  |  |  |  |  |
| ASET1 | 123456 | 7   | THRU   | 109 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification numbers. (Integer > 0; for THRU option, ID1 < ID2)   |

Remarks:

1. Degrees-of-freedom specified on this entry form members of the a-set that is exclusive from other sets defined by Bulk Data entries. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. If the alternate format is used, all points in the sequence ID1 through ID2 are not required to exist, but there must be at least one degree-of-freedom in the a-set for the model, or a fatal error will result. Any points implied in the THRU that do not exist will collectively produce a warning message but will otherwise be ignored.
4. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

**ATBACC****Acceleration Field applied to ATB Segments- SOL 700**

Defines an acceleration field that will be applied to ATB segments. Used in SOL700 only.

**Format:**

| 1      | 2     | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|--------|-------|-------|-------|-------|-------|-------|-------|-------|----|
| ATBACC | LID   |       | SCALE | NX    | NY    | NZ    |       |       |    |
| +      | NAME1 | NAME2 | NAME3 | NAME4 | NAME5 | NAME6 | NAME7 | NAMEi |    |

**Example:**

|        |    |    |         |     |     |     |     |   |  |
|--------|----|----|---------|-----|-----|-----|-----|---|--|
| ATBACC | 32 |    | 386.088 | 1.0 | 0.0 | 0.0 | 0   | 0 |  |
| +      | LT | MT | UT      | N   | H   | RUL | RLL |   |  |

| Field      | Content  |
|------------|--|
| LID        | Number of a set of loads. (Integer > 0; required)  |
| SCALE      | ATBACC scale factor. (Real $\geq 0.0$ ; default=1.0)   |
| NX, NY, NZ | Components of gravity vector. At least one component must be nonzero. (Real; default=0.0)                      |
| NAMEi      | Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required) |

**Remarks:**

1. The acceleration is defined as:

$$\alpha(t) = T(t).SCALE. N$$

where SCALE is the acceleration scale factor; is the vector defined by NX, NY, and NZ; is the value interpolated at time from the table referenced by the TLOADn entry.

2. LID must be referenced by a TLOADn entry.
3. The type field on the TLOADn entry must be set to zero.
4. More than one ATBACC acceleration field can be defined per problem.
5. This acceleration field is intended to apply a crash pulse to ATB segments that define a crash dummy. The acceleration is multiplied by the mass of the segment and the resulting force is added as an external force.
6. To compare the accelerations of the ATB segments to experiments, the crash pulse needs to be subtracted from the total acceleration. The acceleration of the segments as defined on the H1 entries in the ATB input file are automatically corrected.

**ATBJNT****Interface to ATB Joints- SOL 700**

This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor. Used in SOL700 only.

**Format:**

| 1      | 2  | 3    | 4  | 5  | 6    | 7    | 8    | 9 | 10 |
|--------|----|------|----|----|------|------|------|---|----|
| ATBJNT | ID | NAME |    |    |      |      |      |   |    |
| +      | G0 | G1   | G2 | G3 | EID1 | EID2 | EID3 |   |    |
| +      | G4 | G5   | G6 | G7 | EID4 | EID5 | EID6 |   |    |

**Example:**

|        |      |      |      |      |      |      |      |  |  |
|--------|------|------|------|------|------|------|------|--|--|
| ATBJNT | 1    | HN   |      |      |      |      |      |  |  |
| +      | 1010 | 1011 | 1012 | 1013 | 1004 | 1005 | 1006 |  |  |
| +      | 2010 | 2011 | 2012 | 2013 | 2004 | 2005 | 2006 |  |  |

| Field | Content  |  |
|-------|--|--|
| LID   | Unique ATBJNT number. (Integer > 0; required)  |  |
| NAME  | Name of an ATB joint as given in the first field of a B.3 entry in the ATB input file. (Character; required)   |  |
| G0-G7 | An ATB joint connects two segments. A local joint coordinate system is attached to each of these two segments. The position and orientation of these two coordinate systems relative to the segment coordinate systems is given on entry B.3 in the ATB input file. For each joint (J = 1, NJNT) a B.3 entry is defined in the ATB input file. The joint J connects the segment JNT(J) as given on the B.3 entry and the segment J + 1. SOL700 finds the two segments that are connected by the joint with name = NAME. The grid points G0-G3 and G4-G7 define the joint coordinate systems for the segments JNT(J) and J + 1, respectively. (Integer > 0; required) |  |
| G0    | located at the origin of the joint coordinate system for the ATB segment JNT (J)   |  |
| G1    | located on the local x-axis.   |  |
| G2    | located on the local y-axis.   |  |
| G3    | located on the local z-axis.   |  |
| G4    | located at the origin of the joint coordinate system for the ATB segment J + 1.  |  |

| Field | Content                      |
|-------|------------------------------|
| G5    | located on the local x-axis. |
| G6    | located on the local y-axis. |
| G7    | located on the local z-axis. |

**Remarks:**

All elements related to an ATB segment refer to the same material number. This material number is defined on the ATBSEG entry. If the material is defined to be rigid by means of a MATRIG entry, all elements can be easily connected to the contact ellipsoid of the ATB segment by means of METHOD=RELLIPS in BCTABLE or BCONPRG entry referencing the MATRIG entry. In this way, all elements related to an ATB segment move together with the ATB segment during the analyses and can be postprocessed.

**ATBSEG****Interface to ATB Segments- SOL 700**

Defines the position and orientation of the ATB segments. The position and orientation as specified on the G.2 and G.3 entries in the ATB input file will be overruled by the definitions given here.

This entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment, together with the contact ellipsoid and the joints it is connected by. See also ATBJNT and DYPARAM, ATBSEG. Used in SOL700 only.

**Format:**

| 1      | 2  | 3    | 4     | 5      | 6      | 7      | 8    | 9      | 10 |
|--------|----|------|-------|--------|--------|--------|------|--------|----|
| ATBSEG | ID | NAME | COVER | NUMELM | GSTART | ESTART | MID  | PIDCOV |    |
| +      | G0 | G1   | G2    | G3     | EID1   | EID2   | EID3 | PIDCG  |    |

**Example:**

|        |      |      |      |      |      |      |      |      |  |
|--------|------|------|------|------|------|------|------|------|--|
| ATBSEG | 1    | HEAD | YES  | 100  | 1000 | 1000 | 1000 |      |  |
| +      | 1010 | 1011 | 1012 | 1013 | 1001 | 1002 | 1003 | 1001 |  |

| Field | Content   |  |
|-------|---|--|
| ID    | Unique ATBSEG number. (Integer > 0; required)   |  |
| NAME  | Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required)  |  |
| G0~G3 | The grid points span the local coordinate system of the ATB segment. G0~G3 are used by SOL700 to overwrite the initial position and orientation of the segments as specified in the ATB input file. EID1-EID3 are about how to generate the grid points for an existing ATB input file. (Integer > 0; required) |  |
| G0    | located at the origin of the ATB segment.   |  |
| G1    | located on the local x-axis.  |  |
| G2    | located on the local y-axis.  |  |
| G3    | located on the local z-axis.  |  |

**AXIC****Conical Shell Problem Flag**

Defines the existence of an axisymmetric conical shell problem.

**Format:**

| 1    | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|------|---|---|---|---|---|---|---|---|----|
| AXIC | H |   |   |   |   |   |   |   |    |

**Example:**

|      |    |  |  |  |  |  |  |  |  |
|------|----|--|--|--|--|--|--|--|--|
| AXIC | 15 |  |  |  |  |  |  |  |  |
|------|----|--|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| H          | Highest harmonic defined for the problem. (0 ≤ Integer < 998) |

**Remarks:**

1. Only one AXIC entry is allowed. When the AXIC entry is present, most other entries are not allowed.  
The types that are allowed with the AXIC entry are listed below:

|         |         |         |
|---------|---------|---------|
| CCONEAX | MATT1   | SPCADD  |
| DAREA   | MOMAX   | SPCAX   |
| DELAY   | MOMENT  | SUPAX   |
| DLOAD   | MPCADD  | TABDMP1 |
| DMI     | MPCAX   | TABLED1 |
| DMIG    | NOLIN1  | TABLED2 |
| DPHASE  | NOLIN2  | TABLED3 |
| EIGB    | NOLIN3  | TABLED4 |
| EIGC    | NOLIN4  | TABLE11 |
| EIGP    | NSM     | TABLEM2 |
| EIGR    | NSM1    | TABLEM3 |
| EIGRL   | NSMADD  | TABLEM4 |
| EPOINT  | OMITAX  | TEMPAX  |
| FORCE   | PARAM   | TF      |
| FORCEAX | PCONEAX | TIC     |
| FREQ    | POINTAX | TLOAD1  |
| FREQ1   | PRESAX  | TLOAD2  |
| FREQ2   | RINGAX  | TSTEP   |

|      |        |
|------|--------|
| GRAV | RFORCE |
| LOAD | RLOAD1 |
| MAT1 | RLOAD2 |
| MAT2 | SECTAX |

2. For a discussion of the conical shell element, see the [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

**AXIF****Fluid Related Axisymmetric Parameters**

Defines basic parameters and the existence of an axisymmetric fluid analysis.

**Format:**

| 1    | 2   | 3  | 4    | 5  | 6     | 7      | 8 | 9 | 10 |
|------|-----|----|------|----|-------|--------|---|---|----|
| AXIF | CID | G  | DRHO | DB | NOSYM | F      |   |   |    |
|      | N1  | N2 | N3   | N4 | N5    | -etc.- |   |   |    |

**Example:**

|      |   |      |      |       |     |  |   |    |  |
|------|---|------|------|-------|-----|--|---|----|--|
| AXIF | 2 | 32.2 | 0.12 | 2.4+5 | YES |  |   |    |  |
|      | 1 | 2    | 3    |       | 4   |  | 7 | 10 |  |

**Alternate Formats and Examples of Continuation Data:**

|  |    |        |    |  |  |  |  |  |  |
|--|----|--------|----|--|--|--|--|--|--|
|  | N1 | "THRU" | Ni |  |  |  |  |  |  |
|  | 0  | THRU   | 10 |  |  |  |  |  |  |

|  |    |        |    |        |    |  |  |  |  |
|--|----|--------|----|--------|----|--|--|--|--|
|  | N1 | "THRU" | Ni | "STEP" | NS |  |  |  |  |
|  | 0  | THRU   | 9  | STEP   | 3  |  |  |  |  |

|      |     |        |    |      |    |    |  |  |  |
|------|-----|--------|----|------|----|----|--|--|--|
| AXIF | 100 | -386.0 |    | 0.0  | NO |    |  |  |  |
|      | 0   | THRU   | 50 | STEP | 5  |    |  |  |  |
|      | 52  |        |    |      |    |    |  |  |  |
|      | 54  | THRU   | 57 |      |    |    |  |  |  |
|      | 61  | THRU   | 65 |      |    |    |  |  |  |
|      | 68  |        | 71 |      | 72 | 75 |  |  |  |
|      | 81  | 92     |    |      |    |    |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| CID        | Fluid coordinate system identification number. (Integer > 0)                      |
| G          | Value of gravity for fluid elements in the axial direction. (Real)                |
| DRHO       | Default mass density for fluid elements. (Real > 0.0 or blank)                    |
| DB         | Default bulk modulus for fluid elements. (Real)                                   |
| NOSYM      | Request for nonsymmetric (sine) terms of series. (Character: "YES" or "NO")       |
| F          | Flag specifying harmonics. (Blank if harmonic is specified, or Character: "NONE") |

| Descriptor | Meaning   |
|------------|---|
| Ni         | Harmonic numbers for the solution, represented by an increasing sequence of integers.<br>On continuation entries, without the “THRU” option, blank fields are ignored.<br>“THRU” implies all numbers including upper and lower harmonics.<br>( $0 \leq \text{Integer} < 100$ , or Character: “THRU”, “STEP” or blank) |
| NS         | Every NSth step of the harmonic numbers specified in the “THRU” range is used for solution. If field 5 is “STEP”, $\text{Ni} = i^*NS+N1$ where $i$ is the number of harmonics.<br>(Integer)   |

**Remarks:**

1. Only one AXIF entry is allowed.
2. CID must reference a cylindrical or spherical coordinate system.
3. Positive gravity (+G) implies that the direction of free fall is in the -Z direction of the fluid coordinate system.
4. The DRHO value replaces blank values of RHO on the FSLIST, BDYLIST and CFLUIDi entries.
5. The DB value replaces blank values of B on the CFLUIDi entries. If the CFLUIDi entry is blank and DB is zero or blank, the fluid is incompressible.
6. If NOSYM = “YES”, both sine and cosine terms are specified. If NOSYM = “NO”, only cosine terms are specified.
7. If F = “NONE”, no harmonics are specified, no fluid elements are necessary, and no continuations may be present. In this case, AXIS = “FLUID” should not be specified in the Case Control Section.
8. Superelements cannot be used.

**AXSLOT****Axisymmetric Slot Analysis Parameters**

Defines the harmonic index and the default values for acoustic analysis entries.

**Format:**

| 1      | 2    | 3  | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|--------|------|----|---|----|----|---|---|---|----|
| AXSLOT | RHOD | BD | N | WD | MD |   |   |   |    |

**Example:**

|        |       |       |   |      |   |  |  |  |  |
|--------|-------|-------|---|------|---|--|--|--|--|
| AXSLOT | 0.003 | 1.5+2 | 3 | 0.75 | 6 |  |  |  |  |
|--------|-------|-------|---|------|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| RHOD       | Default density of fluid in units of mass/volume. (Real > 0.0 or blank)                         |
| BD         | Default bulk modulus of fluid in units of force/volume ratio change. (Real $\geq$ 0.0 or blank) |
| N          | Harmonic index number. (Integer $\geq$ 0 or blank)  |
| WD         | Default slot width. (Real $>$ 0.0 or blank)   |
| MD         | Default number of slots. (Integer $\geq$ 0 or blank)  |

**Remarks:**

1. Only one AXSLOT entry is allowed.
2. If any of the RHO, B, W, and M fields on the GRID, SLBDY, CAXIFI, and CSLOTi entries are blank, then values must be specified for the RHOD, BD and MD fields.
3. If the number of slots (M) is different in different regions of the cavity, this fact may be indicated on the CSLOTi and SLBDY entries. If the number of slots is zero, no matrices for CSLOTi elements are generated.
4. BD= 0.0 implies the fluid is incompressible.

**BAROR****CBAR Entry Default Values**

Defines default values for field 3 and fields 6 through 8 of the CBAR entry.

**Format:**

| 1     | 2 | 3   | 4 | 5 | 6  | 7  | 8  | 9    | 10 |
|-------|---|-----|---|---|----|----|----|------|----|
| BAROR |   | PID |   |   | X1 | X2 | X3 | OFFT |    |

**Example:**

|       |  |    |  |  |     |     |       |     |  |
|-------|--|----|--|--|-----|-----|-------|-----|--|
| BAROR |  | 39 |  |  | 0.6 | 2.9 | -5.87 | GOG |  |
|-------|--|----|--|--|-----|-----|-------|-----|--|

**Alternate Format and Example:**

|       |  |     |  |  |    |  |  |      |  |
|-------|--|-----|--|--|----|--|--|------|--|
| BAROR |  | PID |  |  | G0 |  |  | OFFT |  |
| BAROR |  | 39  |  |  | 18 |  |  | GOG  |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number of the PBAR entry. (Integer > 0 or blank)  |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)  |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ , using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBAR entry) |
| OFFT       | Offset vector interpretation flag. See Remark 5. (Character or blank)   |

**Remarks:**

1. The contents of fields on this entry will be assumed for any CBAR entry whose corresponding fields are blank.
2. Only one BAROR entry is allowed.
3. For an explanation of bar element geometry, see the [Three-Node Beam Element \(CBEAM3\)](#) in the *MSC Nastran Reference Guide*.
4. If field 6 is an integer, then G0 is used to define the orientation vector and X2 and X3 must be blank. If field 6 is real or blank, then X1, X2, and X3 are used.
5. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

| String | Orientation Vector | End A Offset | End B Offset |
|--------|--------------------|--------------|--------------|
| GGG    | Global             | Global       | Global       |
| BGG    | Basic              | Global       | Global       |
| GGO    | Global             | Global       | Offset       |
| BGO    | Basic              | Global       | Offset       |
| GOG    | Global             | Offset       | Global       |
| BOG    | Basic              | Offset       | Global       |
| GOO    | Global             | Offset       | Offset       |
| BOO    | Basic              | Offset       | Offset       |

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character “O” in the table replaces the obsolete character “E”.)

**BARRIER**

## Barrier for Eulerian Transport

Defines a barrier for transport in a Eulerian mesh. Used in SOL 700 only.

Format:

| 1       | 2    | 3    | 4    | 5    | 6      | 7    | 8 | 9 | 10 |
|---------|------|------|------|------|--------|------|---|---|----|
| BARRIER | BID  | BCID | MESH | DIR  | SKFRIC |      |   |   |    |
|         | XMIN | XMAX | YMIN | YMAX | ZMIN   | ZMAX |   |   |    |

Example:

|         |     |    |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|
| BARRIER | 100 | 20 |  |  |  |  |  |  |  |
|         |     |    |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Unique barrier number. (Integer $> 0$ ; Required)   |
| BCID       | Number of a set of BCSEG entries that define the element faces that are barriers to Eulerian transport. See Remark 4. (Integer $\geq 0$ )   |
| MESH       | Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 5. (Integer $\geq 0$ )   |
| DIR        | Allowed values are NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 6. (Character)   |
| SKFRIC     | Skin friction value. See Remark 8. (Real $\geq 0.0, 0.0$ )  |
| XMIN-ZMAX  | Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 7. (Real) |

Remarks:

1. Material cannot pass through any of the faces referenced.
2. Barriers can be modeled on the edge as well as the inside of an Eulerian mesh.
3. BARRIER overrules FLOW and FLOWT definition.
4. BCID is optional. If used, all other inputs are ignored. If not used, the barrier can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
5. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
6. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.

7. XMIN, XMAX, YMIN, etc are only used when both BCID and DIR are blank. The XMIN,YMIN option defines an area on the MESH BOX boundary as shown in [Figure 9-1](#). If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1e+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.

8. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here  $\tau_w$  denotes the shear friction in an Euler element adjacent to a couple surface segment where  $\rho$  is the density and  $u$  is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment. SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

9. Internal barriers can only be defined on existing Euler element faces using the BCID option. They cannot be defined by using the XMIN,YMIN option.

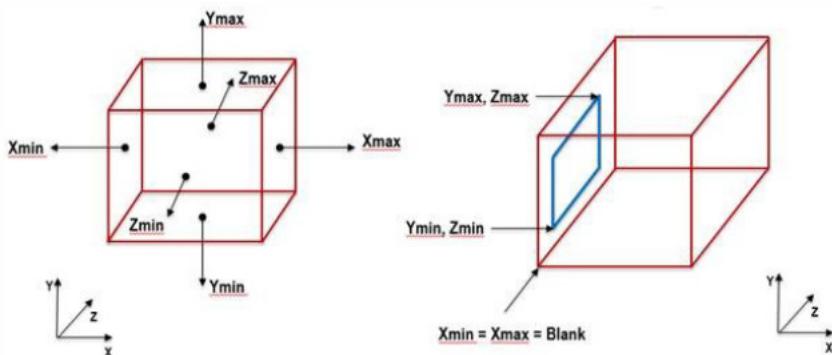


Figure 9-1      MESH BOX boundary

**BCAUTOP**

Automatic contact generation property.

Specifies parameters for automatic contact generation (ACG). With ACG, the code automatically generates the contact bodies based on the grids and elements given, then establishes the contact pairs in which two contact bodies are in contact or may come into contact potentially based on the DISTANCE tolerance. Please refer to the section of [Chapter 9: Contact](#), SOL 400 User's Guide for details.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| BCAUTOP | PID    |        | Param1 | Value1 | Param2 | Value2 | Param3 | Value3 |    |
|         | Param4 | Value4 | etc    |        |        |        |        |        |    |

**Example:**

|         |  |  |       |       |          |     |  |  |  |
|---------|--|--|-------|-------|----------|-----|--|--|--|
| BCAUTOP |  |  | CTYPE | PGLUE | DISTANCE | 0.1 |  |  |  |
|---------|--|--|-------|-------|----------|-----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Reserved for future use (integer $\geq 0$ , default is 0)                     |
| Param(i)   | Name of a parameter. Allowable names are given below (Character).             |
| Value(i)   | Value of the parameter. See below (Real, Integer or character). See Remark 3. |

**Remarks:**

1. Only one entry of BCAUTOP is allowed.
2. This entry does not have effect if BCONTACT=AUTO is not present in case control section.
3. The parameters in this entry may be divided into two types – primary parameters and secondary parameters. The primary parameters are the most important parameters for build of contact bodies, match of contact pairs. The secondary parameters are helpful for user to provide the more detail and further requirement for contact model establishment and contact analysis with automatic contact generation. The primary and secondary parameters are listed in the tables as follows.

Table 9-1 Primary Parameters in BCAUTOP

| Descriptor | Meaning   |
|------------|---|
| CTYPE      | Characters, one of TOUCH(default), PGLUE, SGLUE and GGLUE. If CTYPE is specified in both BCAUTOP entry and Case Control Command BCONTACT=AUTO, the specification in BCAUTOP entry will be used.   |
| DISTANCE   | Distance tolerance of contact pair. If the distance between any two points which are belonging to two different contact bodies is less than this value, these two contact bodies are recognized to be a contact pair. Default value of DISTANCE is 100 times of contact ERROR tolerance. ERROR may be defined in either BCAUTOP or left blank.<br><br>With default of ERROR, the code calculates ERROR as the smallest one of the following values: |
|            | <ol style="list-style-type: none"> <li>1. 1/20 of the smallest nonzero element dimension (plates or solids) in the contact body;</li> <li>2. 1/4 of the thinnest shell thickness in the contact body.</li> </ol>  |
|            | Please see BCPARA for ERROR definition in detail.   |
| ESET       | ID of SET1 to specify element ids consisting contact bodies. If it is not specified, all elements in the model will be used. Only one of ESET, SeedESet, SeedGSet and PropSet can be specified in the same entry.   |
| SeedESet   | ID of SET1 to specify element ids as seeds of contact bodies. The program searches elements as contact bodies from the seed elements. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.   |
| SeedGSet   | ID of SET1 to specify grids ids as seeds of contact bodies. The program searches elements as contact bodies from the elements connecting to the seed grid ids. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.  |
| PropSet    | ID of SET1 to specify property ids to construct contact bodies. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.   |
| RigidSet   | ID of SET1 to specify IDs of rigid bodies to be in contact  |

Table 9-2 Secondary Parameters in BCAUTOP

| Descriptor | Meaning  |
|------------|--|
| BEAMCNT    | Characters, YES or NO (default). Determining if beam is included in contact definition.<br><br>YES: BEAMs are included in the contact description.<br><br>NO: No BEAM is included in the contact description.                                      |
| BEAMCAP    | Characters, FREEEND(default) or NO. If set to FREEEND(default), BCSCAP entries will be added to free end of beam elements, if NO, no BCSCAP will be added. It is mainly used for contact pair detection where the contact occurs at free beam end. |
| DIMENS     | Characters, 2D or 3D (default), dimension of contact bodies.   |
| EDGECNT    | Characters, YES or NO (default). Determining if shell edge is included in contact description.<br><br>YES: The free and hard shell edges are included in the contact definition<br><br>NO: No shell edges are included in the contact definition.  |
| FTYPE      | Characters, BLCOUL/BLSHEAR; or Integer, 6 / 7. BLCOUL or 6 is Bilinear Coulomb friction. BLSHEAR or 7 is Bilinear Shear friction. Default is No Friction.  |
| FRIC       | Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq 0.0$ or Integer $> 0$ ; Default = 0.0)                               |
| IGNTHK     | Character, YES or NO (default), Ignore thickness of shell for contact<br><br>YES: ignore the thickness of the shell<br><br>NO: include the thickness of the shell.   |
| INISTF     | Character, YES or NO (default). Set the option of initial stress free.<br><br>Yes: initial stress free contact<br><br>No: general contact without implementation of initial stress free.   |
| SelfCont   | Characters, YES or NO (default). Option of self-contact.   |

**Note:**

Note that the parameters defined in BCAUTOP are applied only for BONATCT=AUTO. Their default values are defined only when BCONTACT=AUTO is applied. All the parameters in BCPARA, BCONPRP, BCONPRG and BCBDPRP entries may be used in BCAUTOP directly. For the detailed list of these parameters, please refer to BCPARA, BCONPRP, BCONPRG and BCBDPRP.

**BCBDPRP****Contact Body Parameters in SOLs 101 and 400**

Defines contact body parameters used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCBODY1 entry.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| BCBDPRP | PID    |      | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 |    |
|         | PARAM4 | VAL4 | PARAM5 | VAL5 | -etc.- |      |        |      |    |

**Example:**

|         |    |  |      |      |       |   |  |  |  |
|---------|----|--|------|------|-------|---|--|--|--|
| BCBDPRP | 90 |  | FRIC | 0.05 | ISTYP | 0 |  |  |  |
|---------|----|--|------|------|-------|---|--|--|--|

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

PID      Parameter identification number (Integer > 0).

PARAMi      Name of a parameter. Allowable names are given in [Table 9-3](#) (Character).

VALi      Value of a parameter. See [Table 9-3](#) (Real or Integer).

Table 9-3

Contact Body Parameters in SOLs 101 and 400

| Name  | Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)   |
|-------|---|
| BNC   | Exponent associated with the natural convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables.        |
| BNCE  | Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables.              |
| BNL   | Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. |
| BNLE  | Exponent associated with the nonlinear convective heat flow to the environment. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.   |
| CFILM | Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.  |
| CMB   | Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real $\geq 0.0$ ; Default = 0.0) For nonzero values, a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.   |
| CMS   | Heat capacity of the environment, when associated with a scalar point. (Real $\geq 0.0$ ; Default = 0.0) For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.  |
| COPTB | Flag to indicate how body surfaces may contact. See Remark 2. on the <a href="#">BCONPRG</a> entry. (Integer; Default = 0)  |

|       |  |
|-------|--|
| EMISS | Emissivity for radiation to the environment or near thermal radiation. (Real or Integer; Default = 0.0) If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.  |
| FRIC  | <p>Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real <math>\geq 0.0</math> or Integer <math>&gt; 0</math>; Default = 0.0)</p> <p>When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.</p> <p>In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCONECT/BCONPRP is strongly recommended.</p> |
| HBL   | Separation distance dependent thermal convection coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the separation distance dependent thermal convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.   |
| HCT   | Contact heat transfer coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. See Remark 3.   |
| HCV   | Convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.  |
| HNC   | Natural convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.  |
| HNCE  | Natural convection coefficient for heat flow to the environment. (Real or Integer; Default = 0.0) If Real, the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.  |
| HNL   | Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.  |

|       |  |
|-------|--|
| HNLE  | Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real $\geq 0.0$ or Integer; Default = 0.0) If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.   |
| IDSPL | Controls geometric smoothing of boundary of deformable body option. (Integer; Default = 0)<br>=0 or blank, discrete geometric representation.<br>>0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs(IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 1.)<br><0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE. |
| ISTYP | <b>Check</b> of contact conditions. (Integer $\geq 0$ ; Default = 0 for all solutions)<br>ISTYP is not necessary in segment-to-segment contact.<br>For a deformable body:<br>=0 check each body, versus the other.<br>=2 double-sided contact with automatic optimization of contact constraint equations (this option is known as "optimized contact").<br><br>Note that ISTYP is supported with ISEARCH=0 in <a href="#">BCTABLE</a> or <a href="#">BCONPRG</a> only.  |
| ITYPE | An option entry for heat transfer only (Integer; no Default)<br>1 - Heat sink (rigid body)<br>2 - Deformable body (with heat conduction)<br>4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis, i.e. the same contact body with ITYPE=4 cannot be used in mechanical analysis of SOL 400.)   |

|        |   |
|--------|---|
| MIDNOD | Mid-side node projection flag. (Integer $\geq 0$ ; Default = 0)<br>When MIDNOD $> 0$ and IDSPL $\neq 0$ , the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact. Only used if IDSPL is not zero.   |
| SANGLE | Threshold for automatic discontinuity detection in degrees. (Real; Default = 60.0) Used for geometric smoothing option in SOL 400 only. SANGLE is not used and is always set to 0.0 when IDSPL $\geq 0$ .   |
| TBODY  | Body temperature. (Real or Integer; Default = 0.0) If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer, its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8.   |
| TSINK  | Environment sink temperature. (Real or Integer, Default = 0.0) If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8. |

**Remarks:**

1. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See *MSC Nastran Nonlinear User's Guide*, Chapter 9, Contact for more details.
2. For hard contact, with HGLUE=1 (see BCONPRP for the meaning of HGLUE):
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
  - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

3. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

It is recommended to enter the HCT value on the BCONPRP Bulk Data entry, since it generally applies to body pairs and is not a property of a single body.

#### 4. For near contact:

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_B^4) +$$

$$\left[ HCT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] (T_A - T_B)$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCONPRP Bulk Data entry, since they generally apply to body pairs and are not properties of a single body.

#### 5. For no contact:

$$q = CFILM \cdot (T_A - T_{SINK}) +$$

$$HNCE \cdot (T_A - T_{SINK})^{BNCE} +$$

$$HNLE \cdot (T_A^{BNLE} - T_{SINK}^{BNLE}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_{SINK}^4)$$

6. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
7. The  $T_{SINK}$  and  $T_{BODY}$  temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).
8. TBODY entries only apply to rigid bodies (i.e., RIGID as the BEHAV value in BCBODY1 entry). TSINK entries only apply to deformable bodies (meshed regions with DEFORM or HEAT as the BEHAV value in BCBODY1 entry).

**BCBMRAD**

## Modify Equivalent Radius for Beam-to-Beam Contact

Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section in SOL 101, SOL 400 and SOL 600. The BCBMRAD entry is only used for node-to-segment beam-to-beam contact. For segment-to-segment beam contact, the beam cross section geometry is defined via PBARL or PBEAML

**Format:**

| 1       | 2      | 3    | 4   | 5   | 6    | 7   | 8   | 9 | 10 |
|---------|--------|------|-----|-----|------|-----|-----|---|----|
| BCBMRAD | RADIUS | TYPE | ID1 | ID2 | THRU | ID3 | BY  | N |    |
|         | ID4    | THRU | ID5 | ID6 | ID7  | ID8 | ID9 |   |    |

**Example 1:**

|         |     |      |      |      |      |      |      |   |  |
|---------|-----|------|------|------|------|------|------|---|--|
| BCBMRAD | 2.5 | EID  | 100  | 20   | THRU | 300  | BY   | 2 |  |
|         | 200 | 3457 | 8456 | 4712 | 1000 | THRU | 2000 |   |  |
|         | 3.0 |      | 4112 | THRU | 4700 |      |      |   |  |
|         | 2.8 | BODY | 502  | 517  | 3459 |      |      |   |  |

**Example 2:**

|         |     |     |      |      |      |      |  |  |  |
|---------|-----|-----|------|------|------|------|--|--|--|
| BCBMRAD | 2.5 | ALL |      |      |      |      |  |  |  |
|         | 2.8 | EID | 2567 | 1240 | THRU | 1760 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| RADIUS     | Equivalent radius to be used for beam-beam contact problems. See Remark 6. (Real; no Default)                         |
| TYPE       | The attribute of all following ID's. (Character; Default = "EID")   |
| EID        | Defines all the following entries are the IDs of beam-type elements.  |
| BODY       | Defines all the following entries are the IDs of BCBODYs.   |
| ALL        | Defines the default RADIUS for all beam-type elements.  |
| IDi        | ID of a beam-type element, CROD, CBAR, CBEAM and CBEAM3, or a BCBODY with the specified radius. (Integer; no Default) |

**Remarks:**

1. Multiple BCBMRAD Bulk Data entries, which are open-ended are allowed in one file.
2. In each entry of BCBMRAD, there is only one RADIUS input allows, on the field 2. From the field 4 to the rest fields, including all continuation entries, user can input all ID's in any combination of the following 3 basic formats
  - ID1 ID2 ID3 ID4 ...
  - ID1 THRU ID2

- ID1 THRU ID2 BY N

Note that blank fields are allowed for readability.

3. When all beam contact radii are the same, user can put “ALL” on the field 3 following a RADIUS value on the BCBMRAD Bulk Data entry. This value of RADIUS will be applied to all beam-type elements (See the previous Example 2). User can also use this way to give default radius of all beam-type elements.
4. The RADIUS value of the other BCBMRAD's (TYPE=BODY or EID) can override the default value (TYPE=ALL) in the following order:
  - a. When any BCBODY is selected on BCBMRAD with TYPE=BODY, the specified RADIUS will be applied to all beam-type elements in this BCBODY. This value always overrides the default (TYPE=ALL).
  - b. The RADIUS with TYPE=EID always overrides the value from TYPE=BODY and ALL.
5. When TYPE=ALL, then all IDi must be blank.
6. For tubes or round bars, enter the outer radius. For beam, enter an equivalent radius calculated as follows:

$$I = 0.5 \cdot (I_x + I_y)$$

$$R = \sqrt{\frac{A}{p_i^2} + 2 \cdot \frac{I}{A}}$$

7. SOL 600 does not support the TYPE=ALL option.
8. This is only used for node-to-segment contact.

**BCBODY**

## Flexible or Rigid Contact Body in 2D and 3D

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only. The BCBODY1 option is the preferred method to defining contact bodies for SOL 400.

Use only as many forms (i.e. HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The “RIGID” header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Format: (SOLs 101 and 400 only)

| 1      | 2         | 3        | 4                   | 5                       | 6          | 7        | 8       | 9   | 10              |
|--------|-----------|----------|---------------------|-------------------------|------------|----------|---------|---|-----------------|
| BCBODY | BID       | DIM      | BEHAV               | BSID                    | ISTYP      | FRIC     | IDSPL   | CONTROL                                   |                 |
|        | NLOAD     | ANGVEL   | DCOS1               | DCOS2                   | DCOS3      | VELRB1   | VELRB2  | VELRB3                                    |                 |
|        | “ADVANCE” | SANGLE   | COPTB               |                         | MIDNO<br>D |          |         |   |                 |
|        | “RIGID”   | CGID     | NENT                | --- Rigid Body Name --- |            |          |         |   |                 |
|        | “APPROV”  | A        | N1                  | N2                      | N3         | V1       | V2      | V3  |                 |
|        | “GROW”    | GF1      | GF2                 | GF3                     | TAB-GF1    | TAB-GF2  | TAB-GF3 |   |                 |
|        | “HEAT”    | CFILM    | TSINK               | CHEAT                   | TBODY      | HCV      | HNC     | ITYPE                                     |                 |
|        |           | BNC      | EMISS               | HBL                     | HNL        | BNL      | HNLE    | BNLE                                      |                 |
|        |           | HNCE     | BNCE                | CMB                     | CMS        |          |         |   |                 |
|        | “PATCH3D” | NPATCH   |                     |                         |            |          |         |   |                 |
|        |           | IDP      | G1                  | G2                      | G3         | G4       |         |   |                 |
|        |           | IDP      | G1                  | G2                      | G3         | G4       |         |   |                 |
|        |           | etc.     | (npatch<br>entries) |                         |            |          |         |   |                 |
|        | “BEZIER”  | NP1      | NP2                 | NSUB1                   | NSUB2      |          |         |   |                 |
|        |           | G1       | G2                  | G3                      | G4         | etc      |         | (np1*np2<br>values)                       |                 |
|        | “NURBS2D” | NPTU     | NORU                | NSUB                    |            |          |         | (2D<br>Contact)                           |                 |
|        |           | G1 or X1 | G2 or Y1            | G3                      | G4 or X2   | G5 or Y2 | G6      | [abs(nptu)<br>grids or x,<br>y,z, values] | See<br>Remark 8 |

|  |         |          |           |           |          |          |  |                     |              |
|--|---------|----------|-----------|-----------|----------|----------|--|---------------------|--------------|
|  |         | Homo1    | Homo2     | Homo3     | Homo4    | etc.     |  | (nptu values)       |              |
|  |         | Knot1    | Knot2     | Knot3     | Knot4    | Knot5    | etc.   | (nptu+nor u values) |              |
|  | "NURBS" | NPTU     | NPTV      | NORU      | NORV     | NSUBU    | NSUBV  | NTRIM               |              |
|  |         | G1 or X1 | G2 or Y1  | G3 or Z1  | G4 or X2 | G5 or Y2 | G6 or Z2   | G7                  | See Remark 8 |
|  |         | G8 or X3 | G9 or Y3  | G10 or Z3 | etc.     |          | [abs(nptu)<br>*nptv<br>values]                                   | See<br>Remark 8     |              |
|  |         | Homo1    | Homo2     | Homo3     | Homo4    | Homo5    | Homo6  | Homo7               |              |
|  |         | Homo8    | Homo9     | etc.      |          |          | (nptu*npt v ales)  |                     |              |
|  |         | Knot1    | Knot2     | Knot3     | Knot4    | Knot5    | Knot6  | Knot7               | '            |
|  |         | Knot8    | Knot9     | etc.      |          |          | (nptu+nor u+nptv+n orv values)                                   |                     |              |
|  |         | IDtrim   | NPTUtrim  | NORUtrim  | NSUBtrim |          | (repeat<br>this and all<br>following<br>lines<br>NTRIM<br>times) |                     |              |
|  |         |          | Xisoparam | Yisoparam |          |          | (NPTUtri m entries)  |                     |              |
|  |         |          | Homo1     | Homo2     | Homo3    | etc      | (NPTUtri m entries)  |                     |              |
|  |         |          | Knot1     | Knot2     | Knot3    | etc.     | (NPTUtri m +<br>NORUtri m entries)                               |                     |              |

Format: (SOL 700 only)

|        |     |     |       |      |  |  |  |  |  |
|--------|-----|-----|-------|------|--|--|--|--|--|
| BCBODY | BID | DIM | BEHAV | BSID |  |  |  |  |  |
|--------|-----|-----|-------|------|--|--|--|--|--|

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

|        |   |  |        |     |   |     |  |  |  |
|--------|---|--|--------|-----|---|-----|--|--|--|
| BCBODY | 1 |  | DEFORM | 101 | 0 | .05 |  |  |  |
|--------|---|--|--------|-----|---|-----|--|--|--|

Example 2 -- Simple 4-node rigid patch (see Remark 4. for rigid bodies)

|        |         |   |       |     |     |     |  |  |  |
|--------|---------|---|-------|-----|-----|-----|--|--|--|
| BCBODY | 2       |   | RIGID | 102 | 0   | .08 |  |  |  |
|        | PATCH3D | 1 |       |     |     |     |  |  |  |
|        |         | 1 | 101   | 102 | 103 | 104 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)   |
| DIM        | Dimension of body. (Character; Default= 3D) (Ignored by SOL 700)<br>DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves.<br>DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.  |
| BEHAV      | Behavior of curve or surface (Character; Default = DEFORM) (Ignored by SOL 700)<br>DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, HEAT indicates body is a heat-rigid body. <i>See Remark 4. for Rigid Bodies.</i>   |
| BSID       | Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM. For SOL 700 the BSID may also be the identification number of a BCBOX, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0)  |
| ISTYP      | Check of contact conditions. (Integer $\geq$ 0; Default = 0 for all solutions)<br>ISTYP is not necessary in segment-to-segment contact.<br>For a deformable body:<br>=0 check each body, versus the other .<br>=2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”).   |
|            | Note that ISTYP is supported with ISEARCH=0 in <a href="#">BCTABLE</a> or <a href="#">BCONPRG</a> only.   |
| FRIC       | Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq$ 0.0 or Integer > 0; Default = 0.0)<br><br>When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.  |
|            | In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCNECT/BCONPRP is strongly recommended.   |
| IDSPL      | Controls geometric smoothing of boundary of deformable body. (Integer; Default = 0)<br>=0 or blank, discrete geometric representation<br>>0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs (IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 7.)<br><0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE. |

| Descriptor | Meaning  |
|------------|--|
| CONTROL    | = -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRBi in the following line.<br>= 0 for velocity control (default).<br>= positive number for "load control". The positive number entered is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)  |
| NLOAD      | Enter a positive number if "load controlled" and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for rotation about x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively).<br><br>Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation. |
| ANGVEL     | Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)   |
| DCOSi      | Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.  |
| VELRBi     | Translation velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID filed. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0)  |
| "ADVANCE"  | The entries for this continuation line are for advanced options starting with MD Nastran R2.   |
| SANGLE     | Threshold for automatic discontinuity detection in degrees. (Real; Default = 60.0)<br>Used for SPLINE option in SOL 400 only. SANGLE is not used when IDSPL $\geq$ 0.  |
| COPTB      | Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)   |
| MIDNOD     | Mid-side node projection flag. (Integer $\geq$ 0; Default = 0)<br>When MIDNOD > 0 and IDSPL $\neq$ 0, the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact.  |
| "RIGID"    | The entries of this continuation line are for the rigid body description. See Remark 4.  |

| Descriptor            | Meaning   |
|-----------------------|---|
| CGID<br>(5,i) i=1,2,3 | Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied  |
| NENT                  | Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)  |
| Rigid Body Name       | Name associated with the rigid body. (Default is blank, 24-characters maximum)  |
| “APPROV”              | The entries of this continuation line are for approaching velocity to establish initial contact.  |
| A                     | Angular velocity about local axis through center of rotation. (Real, Default = 0.0)   |
| Ni                    | Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the “RIGID”, CGID entry. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)  |
| Vi                    | V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)   |
| “GROW”                | The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2(growth vs time) or TABL3D (growth vs one or more variables).   |
| GFi                   | Components of Growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Real, Default = 1.0)  |
| TAB-GFi               | Tabled IDs for growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Integer > 0 or blank, Default is blank)  |
| “HEAT”                | The entries of this continuation line(s) are for contact in heat transfer in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.   |
| CFILM<br>(9,1)/(10,1) | Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.  |
| TSINK<br>(9,2)/(10,2) | Environment sink temperature. (Real or Integer, Default = 0.0). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry. |

| Descriptor            | Meaning   |
|-----------------------|---|
| CHEAT<br>(9,3)/(10,3) | Contact heat transfer coefficient. (Real or Integer; Default = 0.0). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables.   |
| TBODY<br>(9,4)/(10,4) | Body temperature. (Real or Integer; Default = 0.0). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry.  |
| HCV<br>(9,5)/(10,5)   | Convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.  |
| HNC<br>(9,6)/(10,6)   | Natural convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.   |
| ITYPE<br>[4,8]        | An option entry for heat transfer only (Integer; no Default)<br>1 - Heat sink (rigid body)<br>2 - Deformable body (with heat conduction)<br>4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis)   |
| BNC<br>(9,7)/(10,7)   | Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables. |
| EMISS<br>(9,8)/(10,8) | Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.   |

| Descriptor         | Meaning  |
|--------------------|--|
| HBL<br>(7,6)/(8,6) | Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0). If Real, the value entered is the separation distance dependent thermal convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.  |
| HNL                | Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0). If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.   |
| BNL                | Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. |
| HNLE               | Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real $\geq 0.0$ or Integer; Default = 0.0). If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.  |
| BNLE               | Exponent associated with the nonlinear convective heat flow to the environment (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.    |
| HNCE               | Natural convection coefficient for heat flow to the environment. (Real or Integer, Default = 0.0). If Real the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.  |

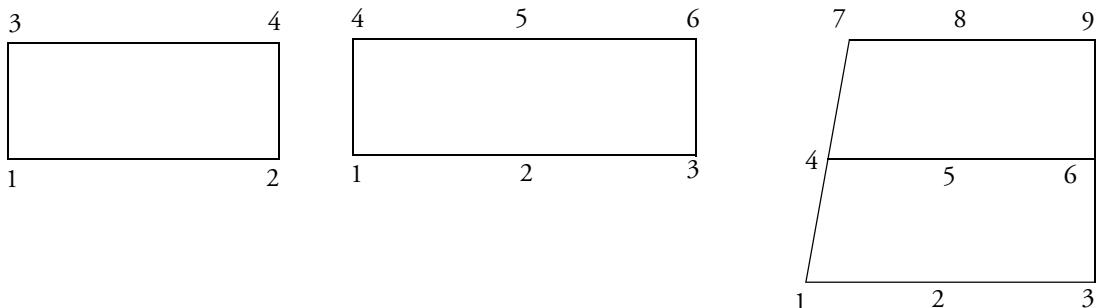
| Descriptor                      | Meaning  |
|---------------------------------|--|
| BNCE                            | Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables. |
| CMB                             | Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real $\geq 0.0$ ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.  |
| CMS                             | Heat capacity of the environment, when associated with a scalar point. (Real $\geq 0.0$ ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.  |
| “PATCH3D”                       | Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)   |
| IDP                             | ID of the patch (Integer number 1 through highest value).  |
| G1, G2, G3, G4                  | Grid numbers for each of the 4 nodes of the patch (see Note 5).  |
| “BEZIER”                        | Entries for this continuation line describe a rigid body made up of Bezier Surfaces.   |
| NP1                             | Number of points in 1st direction. (Integer $> 0$ )  |
| NP2                             | Number of points in 2nd direction. (Integer $> 0$ )  |
| NSUB1                           | Number of subdivisions in 1st direction. (Integer $> 0$ )  |
| NSUB2                           | Number of subdivisions in 2nd direction. (Integer $> 0$ )  |
| G1, G2, G3, etc                 | Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)  |
| “NURBS2D”                       | Entries for this continuation line describe a 2D rigid body made up of nurmbs.   |
| NPTU                            | Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer, no Default)   |
| NORU                            | Order  |
| NSUB                            | Number of subdivisions   |
| G1, G2, G3, G4                  | Grid numbers for each of the NPTU control points   |
| X1, Y1, X2, Y2,<br>etc.         | Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.   |
| Homo1,<br>Homo2,<br>Homo3, etc. | Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.   |

| Descriptor                      | Meaning   |
|---------------------------------|---|
| Knot1, Knot2,<br>Knot3, etc.    | Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.  |
| "NURBS"                         | Entries for this continuation line describe a rigid body made up of nurbs.  |
| NPTU                            | Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer > 0; Required) |
| NPTV                            | Number of control points in V direction. (Integer > 0; Required)  |
| NORU                            | Order along U direction. (Integer > 0; Required)  |
| NORV                            | Order along V direction (Integer > 0; Required)   |
| NSUBU                           | Number of subdivisions in U direction (Integer > 0; Required)   |
| NSUBV                           | Number of subdivisions in V direction (Integer > 0; Required)   |
| NTRIM                           | Number of trimming curves (Integer $\geq 0$ or blank)   |
| G1, G2, G3,<br>etc.             | Grid point IDs defining control points (Integer > 0). There must be NPTU*NPTV entries.  |
| X1, Y1, Z1, X2,<br>Y2, Z2, etc. | Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.  |
| Homo1,<br>Homo2,<br>Homo3, etc  | Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)   |
| Knot1, Knot2,<br>Knot3, etc     | Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)  |
| IDtrim                          | ID of trimming vector. There must be NTRIM of these entries and those entries that follow. (Integer > 0)  |
| NPUTtrim                        | Number of control points for this trimming vector. (Integer > 0)  |
| NORUtrim                        | Order for this trimming vector. (Integer > 0)   |
| NSUBtrim                        | Number of subdivisions for this trimming vector. (Integer > 0)  |
| Xisoparam                       | First coordinate of point in isoparametric space. (Real)  |
| Ysoparam                        | Second coordinate of point in isoparametric space. (Real)   |
| Homo1,<br>Homo2,<br>Homo3, etc  | Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)  |
| Knot1, Knot2,<br>Knot3, etc     | Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)  |

**Remarks:**

1. Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
2. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
3. All continuation lines may be omitted if not required.
4. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
5. For BEZIER surfaces, enter np1\*np2 points in the order shown below:

| Mesh | Normal Order      | Reversed Order    |
|------|-------------------|-------------------|
| 2x2  | 1,2,3,4           | 2,1,4,3           |
| 3x2  | 1,2,3,4,5,6       | 3,2,1,6,5,4       |
| 3x3  | 1,2,3,4,5,6,7,8,9 | 3,2,1,6,5,4,9,8,7 |



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value).
7. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
8. For hard contact, with HGLUE=1 (see BCTABLE for the meaning of HGLUE):
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.

- b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

9. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = \text{CHEAT} \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry. It is recommended to enter the CHEAT value on the BCTABLE Bulk Data input, since it generally applies to body pairs and is not a property of a single body.

10. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BNL} - T_B^{BNL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[ \text{CHEAT} \cdot \left( 1 - \frac{\text{dist}}{DQNEAR} \right) + HBL \cdot \frac{\text{dist}}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry. It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCTABLE Bulk Data input, since they generally apply to body pairs and are not properties of a single body.

11. For no contact:

$$\begin{aligned} q = & CFILM \cdot (T_A - T_{SINK}) + \\ & HNCE \cdot (T_A - T_{SINK})^{BNCE} + \\ & HNLE \cdot (T_A^{BNLE} - T_{SINK}^{BNLE}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_{SINK}^A) \end{aligned}$$

12. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).

13. The  $T_{SINK}$  and  $T_{BODY}$  temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).
14. Table IDs of tables used on the BCBODY and the BCTABLE entry must be unique.
15. TBODY entries only apply to rigid bodies (i.e., RIGID as the BEHAV value). TSINK entries only apply to deformable bodies (meshed regions with DEFORM or HEAT as the BEHAV value).

**BCBODY****Flexible or Rigid Contact Body in 2D and 3D in SOL 600**

Defines a flexible or rigid contact body in 2D or 3D used in SOL 600 only.

**Important Notes for SOL 600:**

1. The 2nd line is required if any of the subsequent lines are to be entered (if this line is blank, enter a "+" in column 1 or Nastran will ignore the entire line).
2. PARAM,MRCONVER,11 is required if CTYPE, APPROV, or GROW headers are entered.
3. PARAM,MRCONTAB,11 is required if any of the variables are described by tables.

Use only as many forms (i.e., HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The "RIGID" header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

**Format:**

| 1         | 2       | 3                | 4                       | 5       | 6       | 7       | 8      | 9                | 10 |
|-----------|---------|------------------|-------------------------|---------|---------|---------|--------|------------------|----|
| BCBODY    | BID     | DIM              | BEHAV                   | BSID    | ISTYP   | FRIC    | IDSPL  | CONTROL          |    |
|           | NLOAD   | ANGVEL           | DCOS1                   | DCOS2   | DCOS3   | VELRB1  | VELRB2 | VELRB3           |    |
| "ADVANCE" | SANGLE  | COPTB            | USER                    |         |         |         |        |                  |    |
| "CTYPE"   | ISMALL  | ITYPE            | IAUG                    | PENALT  | AUGDIST |         |        |                  |    |
| "RIGID"   | CGID    | NENT             | --- Rigid Body Name --- |         |         |         |        |                  |    |
| "APPROV"  | A       | N1               | N2                      | N3      | V1      | V2      | V3     |                  |    |
| "RTEMP"   | G(temp) | Tempr            | T(Tempr)                |         |         |         |        |                  |    |
| "SINK"    | G(sink) | Tsink            | T(Tsink)                |         |         |         |        |                  |    |
| "GROW"    | GF1     | GF2              | GF3                     | TAB-GF1 | TAB-GF2 | TAB-GF3 |        |                  |    |
| "HEAT"    | CFILM   | TSINK            | CHEAT                   | TBODY   | HCV     | HNC     | ITYPE  |                  |    |
|           | BNC     | EMISS            | HBL                     |         |         |         |        |                  |    |
| "PATCH3D" | NPATCH  |                  |                         |         |         |         |        |                  |    |
|           | IDP     | G1               | G2                      | G3      | G4      |         |        |                  |    |
|           | IDP     | G1               | G2                      | G3      | G4      |         |        |                  |    |
|           | etc.    | (npatch entries) |                         |         |         |         |        |                  |    |
| "BEZIER"  | NP1     | NP2              | NSUB1                   | NSUB2   |         |         |        |                  |    |
|           | G1      | G2               | G3                      | G4      | etc     |         |        | (np1*np2 values) |    |
| "POLY"    | NP1     | NP2              |                         |         |         |         |        |                  |    |
|           | G1      | G2               | G3                      | G4      | etc     |         |        | (np1*np2 values) |    |

|  |           |          |           |           |          |          |          |   |              |  |
|--|-----------|----------|-----------|-----------|----------|----------|----------|---|--------------|--|
|  | "CYLIND"  | NSUB     |           |           |          |          |          |   |              |  |
|  |           | Gtop     | Rtop      | Gbottom   | Rbottom  |          |          |   |              |  |
|  | "SPHERE"  | NSUB     |           |           |          |          |          |   |              |  |
|  |           | Gcenter  | Radius    |           |          |          |          |   |              |  |
|  | "LINE"    | NPTS     | Ix        | Iy        | Rx       | Ry       |          | (2D Contact)                                      |              |  |
|  |           | G1       | G2        | G3        | G4       | etc.     |          | (npts values)                                     |              |  |
|  | "ARC"     | NPTS     | MethArc   | Ix        | Iy       | Rx       | Ry       | (2D Contact)                                      |              |  |
|  |           | G1       | G2        | G3        | G4       | etc.     |          | (npts values)                                     |              |  |
|  | "SPLINE"  | NPTS     | Ix        | Iy        | Rx       | Ry       |          | (2D Contact)                                      |              |  |
|  |           | G1       | G2        | G3        | G4       | etc.     |          | (npts values)                                     |              |  |
|  | "NURBS2D" | NPTU     | NORU      | NSUB      | Rx       | Ry       |          | (2D Contact)                                      |              |  |
|  |           | G1 or X1 | G2 or Y1  | G3        | G4 or X2 | G5 or Y2 | G6       | [abs(nptu) grids or x,y,z, values]                | See Remark 8 |  |
|  |           | Homo1    | Homo2     | Homo3     | Homo4    | etc.     |          | (nptu values)                                     |              |  |
|  |           | Knot1    | Knot2     | Knot3     | Knot4    | Knot5    | etc.     | (nptu+noru values)                                |              |  |
|  | "NURBS2"  | IDN      |           |           |          |          |          |   |              |  |
|  | "NURBS"   | NPTU     | NPTV      | NORU      | NORV     | NSUBU    | NSUBV    | NTRIM   |              |  |
|  |           | G1 or X1 | G2 or Y1  | G3 or Z1  | G4 or X2 | G5 or Y2 | G6 or Z2 | G7  | See Remark 8 |  |
|  |           | G8 or X3 | G9 or Y3  | G10 or Z3 | etc.     |          |          | [abs(nptu)*n ptv values]                          | See Remark 8 |  |
|  |           | Homo1    | Homo2     | Homo3     | Homo4    | Homo5    | Homo6    | Homo7   |              |  |
|  |           | Homo8    | Homo9     | etc.      |          |          |          | (nptu*nptv values)                                |              |  |
|  |           | Knot1    | Knot2     | Knot3     | Knot4    | Knot5    | Knot6    | Knot7   | '            |  |
|  |           | Knot8    | Knot9     | etc.      |          |          |          | (nptu+noru+ ntpv+nrv values)                      |              |  |
|  |           | IDtrim   | NPTUtrim  | NORUtrim  | NSUBtrim |          |          | (repeat this and all following lines NTRIM times) |              |  |
|  |           |          | Xisoparam | Yisoparam |          |          |          | (NPTUtrim entries)                                |              |  |
|  |           |          | Homo1     | Homo2     | Homo3    | etc      |          | (NPTUtrim entries)                                |              |  |
|  |           |          | Knot1     | Knot2     | Knot3    | etc.     |          | (NPTUtrim + NORUtrim entries)                     |              |  |

### Examples (of Deformable and Rigid Contact):

#### Example 1 -- Typical deformable body

|        |   |  |        |     |   |     |  |  |  |
|--------|---|--|--------|-----|---|-----|--|--|--|
| BCBODY | 1 |  | DEFORM | 101 | 0 | .05 |  |  |  |
|--------|---|--|--------|-----|---|-----|--|--|--|

Example 2 -- Simple 4-node rigid patch (see Remark 5 for rigid bodies)

|        |         |   |       |     |     |     |  |  |
|--------|---------|---|-------|-----|-----|-----|--|--|
| BCBODY | 2       |   | RIGID |     | 0   |     |  |  |
|        |         |   |       |     |     |     |  |  |
|        | PATCH3D | 1 |       |     |     |     |  |  |
|        |         | 1 | 101   | 102 | 103 | 104 |  |  |

Example 3-- Same as Example 2 except that a user subroutine named motion.f will be employed to specify the rigid body motion.

|        |         |   |       |     |     |     |  |  |
|--------|---------|---|-------|-----|-----|-----|--|--|
| BCBODY | 2       |   | RIGID |     | 0   |     |  |  |
|        |         |   |       |     |     |     |  |  |
|        | ADVANCE |   |       | M   |     |     |  |  |
|        | PATCH3D | 1 |       |     |     |     |  |  |
|        |         | 1 | 101   | 102 | 103 | 104 |  |  |

| Descriptor     | Meaning  |
|----------------|--|
| BID<br>(4,1)   | Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)  |
| DIM            | Dimension of body. (Character; Default = 3D)<br>DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves.<br>DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.   |
| BEHAV<br>(4,8) | Behavior of curve or surface (Character; Default = DEFORM) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, ACOUS indicates an acoustic body, WORK indicates body is a workpiece, HEAT indicates body is a heat-rigid body. <i>See Remark 3. for Rigid Bodies..</i> |
| BSID           | Identification number of a BSURF, BCBOX, BCPROP or BCMATL entry if BEHAV=DEFORM. (Integer > 0)   |

| Descriptor       | Meaning  |
|------------------|--|
| ISTYP<br>(4,3)   | <p>Check of contact conditions. (Integer <math>\geq 0</math>; Default = 0)</p> <p>ISTYP is not supported in segment-to-segment contact.</p> <p>For a deformable body:</p> <ul style="list-style-type: none"> <li>=0 symmetric penetration, double sided contact.</li> <li>=1 unsymmetric penetration, single sided contact. (Integer <math>&gt; 0</math>)</li> <li>=2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”).</li> </ul> <p>Notes: single-sided contact (ISTYP=1) with the contact bodies arranged properly using the contact table frequently runs much faster than ISTYP=2.</p> <p>For a rigid body:</p> <ul style="list-style-type: none"> <li>=0 no symmetry condition on rigid body.</li> <li>=1 rigid body is a symmetry plane.</li> </ul>   |
| FRIC<br>(6,7)    | Friction coefficient. (Real $\geq 0$ or integer; Default = 0)<br>If the value is an integer it represents the ID of a TABL3Di.   |
| IDSPL<br>(4,5)   | <p>Set IDSPL=1 to activate the SPLINE (analytical contact) option for a deformable body and for a rigid contact surface. Set it to zero or leave blank to not have analytical contact. (Integer; Default = 0)</p> <p><u>SOL 600 Options:</u></p> <ul style="list-style-type: none"> <li>=0 or blank, SPLINE option is turned off.</li> <li>=1, The body is smoothed out with splines (2D) or Coons surfaces (3D). If SANGLE is not entered, the default 60.0 degrees is used. IDSPL=1 triggers the Marc SPLINE option with the 3rd field of the 3rd datablock set to 1. If analytical contact changes between increment zero and any subcase, the SANGLE Bulk Data entry is required (see the following SANGLE). SANGLE is placed in the 4th field of the 3rd Mac SPLINE datablock and a value of 1 is placed in the 3rd field of the 3rd datablock.</li> <li>&gt;1, Identification number of a BLSEG entry that lists nodes on edges of the body which are excluded from the SPLINE option. (See Remark 10.)</li> </ul> <p>This option may be used for deformable or rigid contact surfaces. See PARAM,MSPLINC0 to enforce C0 continuity.</p> |
| CONTROL<br>(4,6) | <p>Indicates the type of control for the body.</p> <p>Integer:</p> <ul style="list-style-type: none"> <li>= -1 for position control, = 0 for velocity control.</li> <li>= positive number for “load control” (the positive number is the grid number which has translational forces or SDCP’s are applied. The position of this grid is at the center of rotation given in the CGID field. For velocity controlled surfaces, the velocity of the body must be specified by the VELRBi fields. For displacement controlled surfaces the final displacement can be specified using SPCD at the grid ID specified by CONTROL. CONTROL &gt; 0 is not available for enforced motion using SPCD’s for 2D contact using SOL 600.</li> </ul>   |

| Descriptor      | Meaning   |
|-----------------|---|
| NLOAD<br>(4,7)  | Enter a positive number if “load controlled” and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The position of this grid is at the center of rotation given in the CGID field. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for rotation about x, y, z respectively), or starting with the MSC Nastran 2007 release by dof's 4-6 (for rotation about x, y, z respectively). For versions prior to MSC Nastran 2007 only dof's 1-3 could be used. |
| ANGVEL<br>(6,1) | Angular velocity or angular position about local axis through center of rotation. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.  |
| DCOS1<br>(6,4)  | 3D - First component direction cosine of local axis if ANGVEL is nonzero. (Real)  |
|                 | 2D - First coordinate of initial position of rotation of rigid body.  |
| DCOS2<br>(6,5)  | 3D - Second component direction cosine of local axis if ANGVEL is nonzero. (Real)   |
|                 | 2D - Second coordinate of initial position of rotation of rigid body  |
| DCOS3<br>(6,6)  | 3D - Third component direction cosine of local axis if ANGVEL is nonzero. (Real)  |
|                 | 2D - Not used.  |
| VELRB1<br>(5,4) | 2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 1st direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.   |
| VELRB2<br>(5,5) | 2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 2nd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.   |
| VELRB3<br>(5,6) | 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 3rd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.  |
|                 | 2D - Not used.  |
| ADVANCE         | The entries for this continuation line are for advanced options starting with MSC Nastran 2007.   |
| SANGLE          | Threshold for automatic discontinuity detection in degrees. (Real, Default = 60.0) SANGLE values on the BCBODY entry can be overridden using the SANGLE Bulk Data entry. This allows SANGLE to change for different subcases. It also allows analytical contact for this body to be “on” or “off” for different subcases (see the SANGLE Bulk Data entry). [Marc Spline option (3,4)]   |
| COPTB           | Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)  |

| Descriptor      | Meaning   |
|-----------------|---|
| USER            | <p>A series of character flags to indicate which (if any) user subroutines are required for this contact body. Enter as many characters as necessary to specify the user subroutines desired, for example to use MOTION and UFIRC enter MF. (Character; Default = blank meaning no user subroutines are required) (SOL 600 Only)</p> <p>M = The MOTION user subroutine is required to describe complex motion of a rigid contact body.</p> <p>F = The UFRIC or UFRICBBC user subroutine is required to describe complex friction behavior of the contact body.</p> <p>C = Film coefficient and sink temperature user subroutine UHTCON is required (coupled structural/heat analysis only)</p> <p>P = 4-Node patch user subroutine DIGEOM is required (the coordinates of the patch are entered in the user subroutine rather than on the BCBODY entry).</p> <p>S = The SEPSTR or SEPFOR user subroutine is required depending on whether stress-based or force-based friction has been specified using BCPARA IBSEP.</p> |
| “CTYPE”         | The entries of this continuation line define whether the original node-to-segment or segment-to-segment contact algorithm is used. Also the selection of general or small sliding contact and associated values when segment-to-segment contact is selected are entered here. These options may be placed on BCPARA instead of BCBODY (which is the recommended option. BCBODY entries override BCPARA entries). For MSC Nastran 2012, these items should be considered beta-test capabilities.   |
| ISMALL<br>[3,1] | Enter 0 for general contact or 1 for small sliding, small displacement contact. See Remarks 15 and 16. (Integer; Default = 0)   |
| ICTYPE<br>[3,2] | Enter 0 for node-to-segment contact or 1 for segment-to-segment contact. See Remarks 15 and 16. (Integer; Default = 0)  |
| IAUG<br>[3,3]   | <p>This entry applies only if segment-to-segment contact is selected. See Remarks 15 and 16. (Integer; Default = 0)</p> <p>0 = No augmentation</p> <p>1 = Augmentation is based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements.</p> <p>2 = Augmentation is based on a constant Lagrange multiplier field.</p> <p>3 = Augmentation is based on a bilinear Lagrange multiplier field.</p>   |

| Descriptor                     | Meaning  |
|--------------------------------|--|
| PENALT<br>[5,1]                | Augmented Lagrange penalty factor, used by segment-to-segment contact only. The default is derived from the contact tolerance distance and the stiffness of the deformable contact bodies or average stiffness if two deformable bodies are in contact. See Remark 18. (Real; Default = leave blank) |
| AUGDIST<br>[5,2]               | Penetration distance beyond which an augmentation will be applied, used by segment-to-segment contact only. This default is 1% of the contact tolerance distance. See Remark 18. (Real; Default = leave blank)   |
| "RIGID"                        | The entries of this continuation line are for the rigid body description. See Remark 3.  |
| CGID<br>(5,i) i=1,2,3<br>(4,6) | Grid point identification number defining the initial position of the center of rotation for the rigid body or the point where a concentrated force or moment is applied.  |
| NENT<br>(4,2)                  | Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)                                   |
| Rigid Body Name<br>(4,9)       | Name associated with the rigid body. (Default is blank, 24-characters maximum)   |
| "APPROV"                       | The entries for this continuation line are used for rigid contact body approach velocity and angular velocity. (For SOL 600, APPVEL may be used instead of APPROV if desired.)   |
| A                              | Angular velocity about local axis through center of rotation. The local axis is defined by DCOS1, DCOS2, DCOS3. (Real; Default = 0.0)  |
| Ni                             | Not used by SOL 600. The direction cosines for approach velocity in SOL 600 are DCOS1, DCOS2, DCOS3 on the second BCBODY line. (Real; Default = 0.0; Ni values entered on this line are ignored for SOL 600)   |
| Vi                             | V1, V2 and V3 define the three components of approach velocity. For 2D contact only two should be entered as defined by Rx or Ry (see each 2D rigid contact body description for the meaning of Rx and Ry). (Real; Default = 0.0)  |
| "RTEMP"                        | The entries of this continuation line describe rigid contact surface temperature variations.   |
| G(Temp)                        | Grid point identification number defining the location of the temperature. (Integer; no Default)   |
| Tempr                          | Temperature at the start of the run. (Real; no Default)  |
| T(Tempr)                       | Identification number of a TABL3Di describing the variation of the rigid body temperature during the analysis. (Integer; Default = 0 which means a constant value Ttempr is used during the entire analysis)   |

| Descriptor            | Meaning   |
|-----------------------|---|
| <b>“SINK”</b>         | The entries of this continuation line describe sink temperatures and their variation for flexible or heat transfer bodies.  |
| G(sink)               | Grid point identification number of defining the location of the sink temperature. (Integer; no Default)  |
| Tsink                 | Sink temperature at the start of the run. (Real; no Default)  |
| T(Tsink)              | Identification number of a TABL3Di describing the variation of the sink temperature during the analysis. (Integer; Default = 0 which means a constant value Tsink is used during the entire analysis)   |
| <b>“GROW”</b>         | The entries for this continuation line are for rigid body growth - This line is rarely used. If entered PARAM,MRCONVER,11 must be used. If tables are used for growth, they should either be TABLED1 (growth vs time) or TABL3Di (growth vs one or more variables) and PARAM,MRCONTAB,11 must be entered.   |
| GF1<br>(7,5)          | Growth factor of rigid body in first coordinate direction. (Real; Default = 1.0)  |
| GF2<br>(7,6)          | Growth factor of rigid body in second coordinate direction. (Real; Default = 1.0)   |
| GF3<br>(7,7)          | Growth factor of rigid body in third coordinate direction. (Real; Default = 1.0)  |
| TAB-GF1<br>(8,5)      | Table for growth factor of rigid body in first coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)  |
| TAB-GF2<br>(8,6)      | Table for growth factor of rigid body in second coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)   |
| TAB-GF3<br>(8,7)      | Table for growth factor of rigid body in third coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)  |
| <b>“HEAT”</b>         | The entries of this continuation line(s) are for contact in heat transfer. Do not enter these line(s) for structural analyses.  |
| CFILM<br>(9,1)/(10,1) | Heat transfer coefficient (film) to environment. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the heat transfer coefficient vs temperature. This is usually called HCVE in the Marc documentation. |

| Descriptor            | Meaning  |
|-----------------------|--|
| TSINK<br>(9,2)/(10,2) | Environment sink temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying temperature vs time. At present, this variable should not be a function of temperature.   |
| CHEAT<br>(9,3)/(10,3) | Contact heat transfer coefficient. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the contact heat transfer coefficient vs temperature.   |
| TBODY<br>(9,4)/(10,4) | Body temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying the body temperature vs time. At present, this variable should not be a function of temperature.  |
| HCV<br>(9,5)/(10,5)   | Convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field convection coefficient vs temperature.  |
| HNC<br>(9,6)/(10,6)   | Natural convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field natural convection coefficient vs temperature.   |
| ITYPE<br>[4,8]        | An option entry for heat transfer only (Integer; no Default)<br>1 - Heat sink<br>4 - Heat conduction body  |
| BNC<br>(9,7)/(10,7)   | Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the exponent associated with the near field natural convection coefficient vs temperature. At present, this variable should not be a function of temperature. |
| EMISS<br>(9,8)/(10,8) | Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 entry specifying the emissivity vs temperature.  |

| Descriptor          | Meaning  |
|---------------------|--|
| HBL<br>(11,1)[12,1] | Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is a convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature. |
| “PATCH3D”           | Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)   |
| IDP                 | ID of the patch (Integer number 1 through highest value).  |
| G1, G2, G3, G4      | Grid numbers for each of the 4 nodes of the patch (see Note 5).  |
| “BEZIER”            | Entries for this continuation line describe a rigid body made up of Bezier Surfaces.   |
| NP1                 | Number of points in 1st direction. (Integer > 0)   |
| NP2                 | Number of points in 2nd direction. (Integer > 0)   |
| NSUB1               | Number of subdivisions in 1st direction. (Integer > 0)   |
| NSUB2               | Number of subdivisions in 2nd direction. (Integer > 0)   |
| G1, G2, G3, etc     | Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)  |
| “POLY”              | Entries for this continuation line describe a rigid body made up of Poly Surfaces.   |
| NP1                 | Number of points in the 1st direction. (Integer > 0)   |
| NP2                 | Number of points in the 2nd direction. (Integer > 0)   |
| G1, G2, G3, etc     | Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)  |
| “CYLIND”            | Entries for this continuation line describe a cylindrical rigid body.  |
| NSUB                | Number of subdivisions. (Integer > 0)  |
| Gtop                | Grid point ID of a grid in the center of the top of the cylinder. (Integer > 0)  |
| Rtop                | Radius of the top of the cylinder. (Real > 0.0)  |
| Gbottom             | Grid point ID of a grid in the center of the bottom of the cylinder. (Integer > 0)   |
| Rbottom             | Radius of the bottom of the cylinder. (Real > 0.0)   |
| “SPHERE”            | Entries for this continuation line describe a spherical rigid body.  |
| NSUB                | Number of subdivisions. (Integer > 0)  |
| Gcenter             | Grid point ID of a grid in the center of the sphere. (Integer > 0)   |
| Radius              | Radius of the sphere. (Real > 0.0)   |
| “LINE”              | Entries for this continuation line describe a 2D rigid body made up of as many line segments as desired.   |

| Descriptor     | Meaning   |
|----------------|---|
| NPTS           | Number of points in the line segment.   |
| Ix             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).  |
| Iy             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2). |
| Rx             | Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)  |
| Ry             | Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)   |
| G1, G2, G3, G4 | Grid numbers for each of the NPTS points on the line segment.   |
| "ARC"          | Entries for this continuation line describe a 2D rigid body made up of as many segments as desired describing an arc.   |
| NPTS           | Number of points in the arc. NPTS must be 4 for an ARC.   |
| MethArc        | Method to generate arc (Integer 0 to 4) (see Marc volume C Contact description (SOL 400 chapter 9) Figures 3-3 and 3-4).                                      |
| Ix             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).  |
| Iy             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2). |
| Rx             | Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)  |
| Ry             | Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)   |
| G1, G2, G3, G4 | Grid numbers for each of the 4 points as described by method.   |
| "SPLINE"       | Entries for this continuation line describe a 2D rigid body made up of as many spline segments as desired.  |
| NPTS           | Number of points for the spline.  |
| Ix             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).  |
| Iy             | Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2). |
| Rx             | Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)  |
| Ry             | Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)   |
| G1, G2, G3, G4 | Grid numbers for each of the NPTS points on the spline.   |

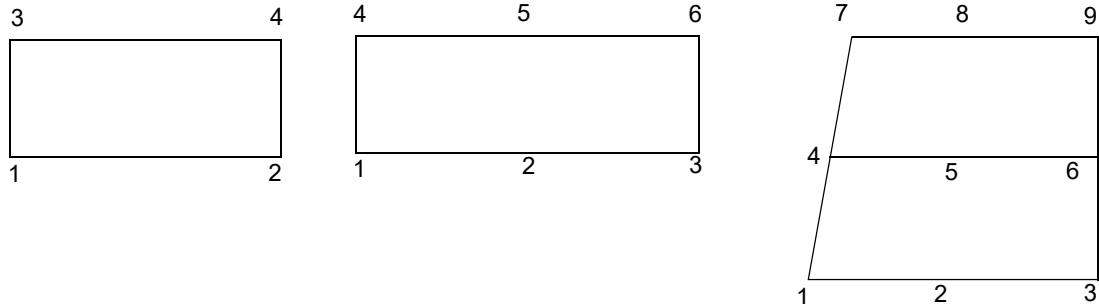
| Descriptor                      | Meaning  |
|---------------------------------|--|
| “NURBS2D”                       | Entries for this continuation line describe a 2D rigid body made up of nurmsbs.  |
| NPTU                            | Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default) |
| NORU                            | Order  |
| NSUB                            | Number of subdivisions   |
| Rx                              | Direction (1, 2, or 3) used as the first VELRBi in Marc’s 2D contact description. (Integer; Default = 1)   |
| Ry                              | Direction (1, 2, or 3) used as the second VELRBi in Marc’s 2D contact description. (Integer; Default = 2)  |
| G1, G2, G3, G4                  | Grid numbers for each of the NPTU control points   |
| X1, Y1, X2, Y2, etc.            | Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) * \text{NPTV}$ (x,y,z) entries.  |
| Homo1, Homo2,<br>Homo3, etc.    | Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.   |
| Knot1, Knot2,<br>Knot3, etc.    | Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.   |
| “NURBS2”                        | Entries for this continuation line describe a rigid body made up of nurbs.   |
| IDN                             | ID of a matching GMNURB entry. The GMNURB is an entry that contains the same information as at shown for the NURBS option. (Integer > 0)   |
| “NURBS”                         | Entries for this continuation line describe a rigid body made up of nurbs.   |
| NPTU                            | Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default) |
| NPTV                            | Number of control points in V direction. (Integer > 0; Required)   |
| NORU                            | Order along U direction. (Integer > 0; Required)   |
| NORV                            | Order along V direction (Integer > 0; Required)  |
| NSUBU                           | Number of subdivisions in U direction (Integer > 0; Required)  |
| NSUBV                           | Number of subdivisions in V direction (Integer > 0; Required)  |
| NTRIM                           | Number of trimming curves (Integer $\geq 0$ or blank)  |
| G1, G2, G3, etc.                | Grid point IDs defining control points (Integer > 0). There must be $\text{NPTU} * \text{NPTV}$ entries.   |
| X1, Y1, Z1, X2, Y2,<br>Z2, etc. | Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) * \text{NPTV}$ (x,y,z) entries.  |

| Descriptor                  | Meaning  |
|-----------------------------|--|
| Homo1, Homo2,<br>Homo3, etc | Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries.<br>(Real)                         |
| Knot1, Knot2,<br>Knot3, etc | Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)                         |
| IDtrim                      | ID of trimming vector. There must be NTRIM of these entries and those entries that follow. (Integer > 0) |
| NPUTtrim                    | Number of control points for this trimming vector. (Integer > 0)   |
| NORUtrim                    | Order for this trimming vector. (Integer > 0)  |
| NSUBtrim                    | Number of subdivisions for this trimming vector. (Integer > 0)   |
| Xisoparam                   | First coordinate of point in isoparametric space. (Real)   |
| Ysoparam                    | Second coordinate of point in isoparametric space. (Real)  |
| Homo1, Homo2,<br>Homo3, etc | Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)     |
| Knot1, Knot2,<br>Knot3, etc | Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)       |

### Remarks:

- Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
- The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
- WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
- (i,j) refers to data block i and field j of Marc's CONTACT model definition entry. IDSPL covers the SPLINE history definition in Marc. For structural analysis (i,j) refers to contact without tables. For heat transfer (i,j) refers to contact with tables.
- For BEZIER surfaces, enter np1\*np2 points in the order shown:

| Mesh | Normal Order      | Reversed Order    |
|------|-------------------|-------------------|
| 2x2  | 1,2,3,4           | 2,1,4,3           |
| 3x2  | 1,2,3,4,5,6       | 3,2,1,6,5,4       |
| 3x3  | 1,2,3,4,5,6,7,8,9 | 3,2,1,6,5,4,9,8,7 |



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value.
7. The heat transfer options are available for SOL 600 starting with MSC Nastran 2005 r2 and must use Marc 2005 or later.
8. For heat transfer items described using a TABLEM1 ID, the smallest value in the table will be entered into Marc's 9th contact (with tables) datablock. The table ID will be translated directly to Marc's 10th contact (with tables) datablock.
9. All flexible surfaces must have smaller BID values than the rigid surfaces so that in the Marc file all flexible surfaces are defined prior to all rigid surfaces. This is a Marc limitation.
10. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
11. With NURBS2D, the coordinates x1, x2, y1, y2 etc. or those given by G1, G2, etc. will be reversed (Y becomes X and X becomes Y) if any CQUADX, CTRIA6 or CTRIA6 entries are found in the bulk data. It is up to the user to ensure that the rigid contact surface is orientated in the correct direction after this reversal. If the user does not want the reversal to happen, enter PARAM,MARCREVX,-1 in the bulk data.
12. Ix and Iy for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIA6 is defined in the Z-X plane, set Ix = 3 and Iy = 1 for all 2D rigid surfaces in the model. For CTRIA6 and CQUADX, normally Ix=2 and Iy=1.
13. Rx and Ry for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIA6 is defined in the Z-X plane, set Rx = 3 and Ry = 1 for all 2D rigid surfaces in the model. For CTRIA6 and CQUADX, normally Rx=2 and Ry=1.
14. For SOL 600, tables using integers for ANGVEL, VELRB1, VELRB2 and/or VELRB3 require PARAM,MRCONVER,11 and PARAM,MRACTOTT,1 to be set. the TABLE3D's used must define the entire motion/velocity desired.
15. If any CTYPE entries are used, PARAM,MRCONVER,11 and PARAM,MRCONVER,11 are required.

16. ISMALL, ITYPE, IAUG should be the same for all BCTABLE entries. The first entry “CTYPE” entry will be used for all bodies. PENALT and AUGDIST may vary from body to body.
17. 2D and 3D rigid bodies may not be mixed in the same input file.
18. The entries on the CTYPE header may optionally be specified on the BCPARA entry.

Correlation between BCBODY “CTYPE” entries and equivalent BCPARA entries:

| BCPARA  | BCPARA        | CTYPE value(s)     |
|---------|---------------|--------------------|
| METHOD  | NODESURF      | ictype=0, ismall=0 |
| METHOD  | NODSMALL*     | ictype=0, ismall=1 |
| METHOD  | SEGSMALL      | ictype=1, ismall=1 |
| METHOD  | SEGMENT*      | ictype=1, ismall=1 |
| METHOD  | SEGLARGE*     | ictype=1, ismall=0 |
| AUTMENT | Integer Value | IAUG=Integer Value |
| PENALT  | Real Value    | PENALT=Real Value  |
| AUGDIST | Real Value    | AUTDIST=Real Value |

\* Not available in SOL 600.

19. Most continuation lines may be omitted if not required except that the second line is required if the third or subsequent lines are to be entered.

**BCBODY1**

## Flexible or Rigid Contact Body in 2D and 3D (New Form)

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only.

If the body is rigid, this entry may refer to BCRIGID entry by the BCRGID field to describe the body. See Remark 1. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM. Unless shrink fit or interference fit is being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

**Format:** (SOLs 101 and 400 only)

| 1       | 2   | 3    | 4   | 5     | 6    | 7      | 8 | 9 | 10 |
|---------|-----|------|-----|-------|------|--------|---|---|----|
| BCBODY1 | BID | BPID | DIM | BEHAV | BSID | BCRGID |   |   |    |

**Format:** (SOL 700 only)

| 1       | 2   | 3 | 4 | 5 | 6    | 7 | 8 | 9 | 10 |
|---------|-----|---|---|---|------|---|---|---|----|
| BCBODY1 | BID |   |   |   | BSID |   |   |   |    |

**Examples of Deformable and Rigid Contact:**

**Example 1 -- Typical deformable body**

|         |    |      |  |        |     |  |  |  |  |
|---------|----|------|--|--------|-----|--|--|--|--|
| BCBODY1 | 94 | 3001 |  | DEFORM | 103 |  |  |  |  |
|---------|----|------|--|--------|-----|--|--|--|--|

**Example 2 -- Simple rigid patch (see Remark 1. for rigid bodies)**

|         |   |      |  |       |     |     |  |  |  |
|---------|---|------|--|-------|-----|-----|--|--|--|
| BCBODY1 | 6 | 3001 |  | RIGID | 206 | 115 |  |  |  |
|---------|---|------|--|-------|-----|-----|--|--|--|

**Descriptor**      **Meaning**

**BID**      Unique contact body identification number referenced by BCNECT, BCHANGE, or BCMOVE. (Integer > 0; Required)

**BPID**      Parameter identification number of a BCBDPRP entry. (Integer > 0 or blank) Ignored in SOL 700. See Remark 2.

**DIM**      Dimension of body. (Character; Default= 3D) Ignored in SOL 700.  
DIM=2D: planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves.

DIM=3D: any 3D body composed of rigid surfaces, shell elements or solid elements.

**BEHAV**      Behavior of curve or surface (Character; Default = DEFORM) Ignored in SOL 700.  
DEFORM body is deformable; RIGID body is rigid (See Remark 1.); SYMM body is a symmetry rigid body; HEAT indicates body is a heat-rigid body (See Remark 3.).

|        |   |
|--------|---|
| BSID   | For SOLs 101 and 400: Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM or HEAT, or identification number of a BCRGSRF, BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry if BEHAV=RIGID or SYMM (See Remark 4.).<br><br>For SOL 700: Identification number (RBID) of a BSURF, BCBOX, BCPROP, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0). |
| BCRGID | For SOLs 101 and 400: Identification number of a BCRIGID entry if BEHAV=RIGID or SYMM. Ignored in SOL 700. (Integer > 0)  |

**Remarks:**

1. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches
2. If BPID field is blank, then default values are set for the parameters defined in BCBDPRP entry.
3. For pure thermal analysis, BEHAV=DEFORM and BEHAV=HEAT are identical. However, in coupled thermal-mechanical analysis, BEHAV=DEFORM specifies a deformable in mechanical analysis part, while BEHAV=HEAT defines a rigid meshed body in mechanical analysis part.
4. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry directly (RBID).

**BCBOX****3D Contact Region**

Defines a 3D contact region - all elements within the region define a contact body in SOL 600 and 700.

**Format (Form 1):**

| 1     | 2  | 3  | 4   | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|----|----|-----|----|----|----|----|----|----|
| BCBOX | ID |    | HOW |    |    |    |    |    |    |
|       | N1 | N2 | N3  | N4 | N5 | N6 | N7 | N8 |    |

**Form 2:**

| BCBOX | ID | COORD | HOW |    |    |    |    |    |  |
|-------|----|-------|-----|----|----|----|----|----|--|
|       |    |       | X1  | Y1 | Z1 | X2 | Y2 | Z2 |  |
|       |    |       | X3  | Y3 | Z3 | X4 | Y4 | Z4 |  |
|       |    |       | X5  | Y5 | Z5 | X6 | Y6 | Z6 |  |
|       |    |       | X7  | Y7 | Z7 | X8 | Y8 | Z8 |  |

**Example (for Form 1):**

|       |      |      |      |      |      |      |      |      |  |
|-------|------|------|------|------|------|------|------|------|--|
| BCBOX | 101  |      | 0    |      |      |      |      |      |  |
|       | 1001 | 1002 | 1003 | 1004 | 1005 | 1006 | 1007 | 1008 |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control BCCONTACT=BCBOX is specified. All elements corresponding to the designated box may potentially come into contact. See Remark 1. (Integer > 0) |
| COORD      | Enter COORD in field 3 if x,y,z coordinates of the box are to be specified rather than grid IDs. (Character)   |
| HOW        | A flag indicating whether an element is in the defined box or not. (Integer; Default = 0)  |
|            | 0 If only one grid point of an element is in the box, the entire element is considered to be in the box.   |
|            | 1 All grid points comprising the element must be within the box, otherwise the element is considered outside of the box.   |
| N1-N8      | Enter 8 Grid IDs defining a box (hexa-like region) if the third field is blank. (Integer; Required if COORD is blank)  |
| Xi, Yi, Zi | Enter eight x,y,z values in the basic coordinate system if the third field is COORD. See Remark 6. (Real; Required if "COORD" is entered in field 3 of line 1)   |

**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCPROP, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. The model is searched to determine whether each element lies within the specified box region as specified by the HOW criteria option.
6. For SOL 700, the BCBOX allows a general box and edges do not need to be aligned with the coordinates axis. The eight grid points define the box identical to the CHEXA grid point numbering.
7. BCBOX is not supported in SOLs 101 and 400.

**BCBZIER**

Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400

Defines a rigid contact body made up of Bezier surfaces used in SOLs 101 and 400 only.

Format:

| 1       | 2    | 3   | 4   | 5     | 6      | 7 | 8 | 9 | 10 |
|---------|------|-----|-----|-------|--------|---|---|---|----|
| BCBZIER | RBID | NP1 | NP2 | NSUB1 | NSUB2  |   |   |   |    |
|         | G1   | G2  | G3  | G4    | -etc.- |   |   |   |    |

Example:

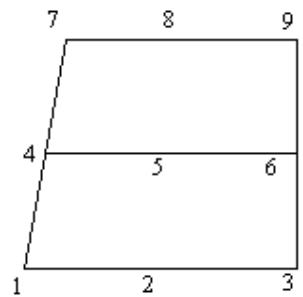
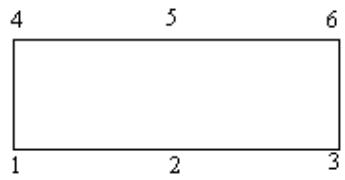
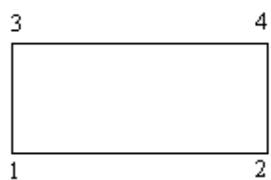
|         |     |    |   |    |    |    |     |     |  |
|---------|-----|----|---|----|----|----|-----|-----|--|
| BCBZIER | 98  | 3  | 3 | 25 | 15 |    |     |     |  |
|         | 21  | 11 | 1 | 84 | 74 | 64 | 147 | 137 |  |
|         | 127 |    |   |    |    |    |     |     |  |

| Descriptor          | Meaning   |
|---------------------|---|
| RBID                | Unique identification number referenced by a BCRGSRF or BCBODY1 entry. (Integer > 0) See Remark 1.  |
| NP1                 | Number of points in 1st direction. (Integer > 0)  |
| NP2                 | Number of points in 2nd direction. (Integer > 0)  |
| NSUB1               | Number of subdivisions in 1st direction. (Integer > 0)  |
| NSUB2               | Number of subdivisions in 2nd direction. (Integer > 0)  |
| G1, G2,<br>G3, etc. | Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP1 points for NP2=2, etc. (Integer > 0) |

Remarks:

1. If BCRGSRF entry does not exist, BCBZIER entry will be referenced by the [BCBODY1](#) entry directly.
2. For BEZIER surfaces, enter np1\*np2 points in the order shown below

| Mesh | Normal Order      | Reversed Order    |
|------|-------------------|-------------------|
| 2x2  | 1,2,3,4           | 2,1,4,3           |
| 3x2  | 1,2,3,4,5,6       | 3,2,1,6,5,4       |
| 3x3  | 1,2,3,4,5,6,7,8,9 | 3,2,1,6,5,4,9,8,7 |



**BCELIPS****Contact Ellipsoid List- SOL 700**

Defines a list of ellipsoid names (character strings) for use of contact analysis. Used in SOL700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5     | 6     | 7     | 8   | 9 | 10 |
|---------|-----|-------|-------|-------|-------|-------|-----|---|----|
| BCELIPS | CID | NAME1 | NAME2 | NAME3 | NAME4 | NAMEi | etc |   |    |

**Example:**

|         |   |     |     |      |       |  |  |  |  |
|---------|---|-----|-----|------|-------|--|--|--|--|
| BCELIPS | 1 | HUB | RIM | HEAD | CHEST |  |  |  |  |
|---------|---|-----|-----|------|-------|--|--|--|--|

| Field | Content   |
|-------|---|
| CID   | Unique identification number of ellipsoid names which can be used for contact.<br>(Integer > 0; required) |
| NAME  | Name of an ellipsoid. (Character; required)   |

**Remarks:**

Use as many continuations as required to define the complete list of names. A blank field terminates the list..

**BCGRID (SOL 700 only)**

Contact Node Region

Grids to be included in contact analyses in SOL 700 only.

**Format:**

| 1      | 2    | 3    | 4     | 5     | 6     | 7     | 8      | 9    | 10 |
|--------|------|------|-------|-------|-------|-------|--------|------|----|
| BCGRID | CID  | GID1 | GID2  | GID3  | GID4  | GID5  | GID6   | GID7 |    |
|        | GID8 | GID9 | GID10 | GID11 | GID12 | GID13 | -etc.- |      |    |

**Example:**

|        |     |      |      |      |     |     |     |     |  |
|--------|-----|------|------|------|-----|-----|-----|-----|--|
| BCGRID | 100 | 12   | 14   | 17   | 121 | 234 | 235 | 270 |  |
|        | 309 | 1001 | THRU | 2000 | BY  | 2   |     |     |  |

| Descriptor        | Meaning   |
|-------------------|---|
| ID                | Unique identification number of a “cloud” of grid points which can be used for contact or RCONN, BJOIN or WALL. (Integer > 0; Required) |
| GID1,<br>GID2,... | Gridpoint ID. THRU indicates a range to be used. BY is the increment to be used within this range. (Integer > 0; Required)              |

**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, BCMATL, or BCSEG entries.
2. BCGRID may only be used for SLAVE body definitions on the BCTABLE entry.
3. As many continuation lines as necessary may be used to define all GRID points used in the definition.

**BCGRID**

Contact region by a list of grid points.

Define a contact region by a grid point list, for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400.

Format:

| 1      | 2    | 3    | 4    | 5      | 6    | 7    | 8    | 9    | 10 |
|--------|------|------|------|--------|------|------|------|------|----|
| BCGRID | BID  | BPID | DIM  |        |      |      |      |      |    |
|        | GID1 | GID2 | GID3 | "THRU" | GID4 | "BY" | GID5 | GID6 |    |
|        | GID7 |      |      |        |      |      |      |      |    |

Example:

|        |    |    |      |     |   |   |  |  |  |
|--------|----|----|------|-----|---|---|--|--|--|
| BCGRID | 2  | 3  | 3D   |     |   |   |  |  |  |
|        | 12 | 21 | THRU | 101 | 3 | 6 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Unique contact face identification number referenced by BCNECT (Integer > 0; Required). See remark 1. |
| BPID       | Parameter identification number of a BCBDRP entry. (Integer > 0 or blank)                             |
| DIM        | Dimension of the contact entry. (Character; Default= "3D"), "3D" or "2D".                             |
| GIDi       | Grid point ids. (Integer > 0)   |

Remarks:

1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.
2. BCGRID entries are able to coexist with BCBODY1 entries, however a BCGRID can only be used to construct a contact pair with a BCSURF in a BCNECT, it is must be a slave and the BCSURF must be a master.
3. This entry only works in node to segment method.
4. The identification of BCGRID cannot be specified in the continuations of a BCNECT entry.

**BCHANGE****Changes Definitions of Contact Bodies**

Changes definitions of contact bodies used in SOL 101 and SOL 400.

**Format:**

| 1       | 2      | 3    | 4  | 5   | 6      | 7    | 8  | 9   | 10 |
|---------|--------|------|----|-----|--------|------|----|-----|----|
| BCHANGE | ID     | TYPE |    |     | IDBOD1 | N1   | N2 | INC |    |
|         | IDBOD2 | N1   | N2 | INC | IDBOD3 | etc. |    |     |    |

**Example:**

|         |     |      |      |   |   |      |      |   |  |
|---------|-----|------|------|---|---|------|------|---|--|
| BCHANGE | 201 | NODE |      |   | 1 | 1001 | 1010 | 1 |  |
|         | 2   | 2001 | 2021 | 2 |   |      |      |   |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number referenced by a BCONTACT or BCHANGE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer $\geq 0$ ; Required) |
| Type       | Type of modification (Character; Required)   |
|            | NODE              Defines nodes of a contact body which may come into contact.   |
|            | EXCLUDE            Excludes 2 node segments in 2D or 4 node patches in 3D.   |
| IDBODi     | Identification number of a contact body, BCBODY or BCBODY1 entry. (Integer $> 0$ )   |
| N1         | Starting grid ID. More than one N1-N2-INC range may be entered for each body, see Remark 7. (Integer $> 0$ ; Default = 1)                            |
| N2         | Ending grid ID.  |
| INC        | Grid ID increment.   |

**Remarks:**

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCHANGE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCHANGE=ID. Note that if BCHANGE Case Control command exists, it always dominates the selection of BCHANGE Bulk Data entries.
2. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
3. BCHANGE is useful only for saving computer time and is not recommended for general usage.
4. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.
5. BCHANGE with the NODE option is not supported in Segment-to-segment contact.

6. NODE and EXCLUDE may not be used simultaneously in the same BCHANGE entry.
7. If TYPE=NODE, the form of N1-N2-INC range has the following rules:
  - a. The format of INC is either blank or integer ( $\geq 0$ ).
  - b. N1-N2-0 or N1-N2-blank represents 2 nodes (N1,N2) where  $0 < N1$  and  $0 < N2$ ,
  - c. N1-N2-INC, INC>0, represent a range input but  $0 < N1 < N2$ .
8. If TYPE=EXCLUDE, the form of N1-N2-INC range has the following rules:
  - a. For node segment, N1-N2-INC is used. If  $0 < N1 < N2$  and INC>0, the range is applied. If INC-blank or 0, it is 2 nodes (N1,N2) input that can be in any order.
  - b. For 4 nodes patch, (N1,N2,N3,N4), 2 sets of range, which have to input in sequence, are required.  
(IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N4,INC2) where IDBOD1=IDBOD2, Ni>0, INC1 and INC2 are ignored.

The following example is for a 4-node patch (100,110,300,200)

| 1       | 2 | 3       | 4   | 5 | 6 | 7    | 8   | 9 | 10 |
|---------|---|---------|-----|---|---|------|-----|---|----|
| BCHANGE | 1 | EXCLUDE |     |   | 1 | -100 | 110 |   |    |
|         | 1 | 300     | 200 |   |   |      |     |   |    |

- c. For 3 nodes patch, (N1,N2,N3), 2 sets of range, which have to input in sequence, are required  
(IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N3,INC2)  
where IDBOD1=IDBOD2,Ni>0, INC1 and INC2 are ignored.

The following example is for a 3-noded patch (132,97,95)

| 1       | 2 | 3       | 4  | 5 | 6 | 7    | 8  | 9 | 10 |
|---------|---|---------|----|---|---|------|----|---|----|
| BCHANGE | 1 | EXCLUDE |    |   | 1 | -132 | 97 |   |    |
|         | 1 | 95      | 95 |   |   |      |    |   |    |

**BCHANGE****Changes Definitions of Contact Bodies in SOL 600**

Changes definitions of contact bodies used in SOL 600.

**Format:**

| 1       | 2      | 3    | 4    | 5   | 6      | 7    | 8  | 9   | 10 |
|---------|--------|------|------|-----|--------|------|----|-----|----|
| BCHANGE | ID     | TYPE | NBOD |     | IDBOD1 | N1   | N2 | INC |    |
|         | IDBOD2 | N1   | N2   | INC | IDBOD3 | etc. |    |     |    |

**Example:**

|         |     |      |      |   |   |      |      |   |  |
|---------|-----|------|------|---|---|------|------|---|--|
| BCHANGE | 201 | NODE | 2    |   | 1 | 1001 | 1010 | 1 |  |
|         | 2   | 2001 | 2021 | 2 |   |      |      |   |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number referenced by a SUBCASE Case Control command. See Remark 2. (Integer $\geq 0$ ; Required)  |
| Type       | Type of modification (Character; Required)   |
|            | NODE              Defines nodes of a contact body which may come into contact.   |
|            | EXCLUDE          Excludes 2 node segments in 2D or 4 node patches in 3D.   |
| NBOD       | Number of bodies to be modified -- must match number of bodies actually entered. More than one N1-N2-INC range may be entered for each body, see Remark 7. See N1 below. (Integer $> 0$ ; Default = 1) |
| IDBODi     | Identification number of a contact body, BCBODY entry. (Integer $> 0$ )  |
| N1         | Starting grid ID.  |
| N2         | Ending grid ID.  |
| INC        | Grid ID increment.   |

**Remarks:**

1. To place an entry in Marc's phase 0, set ID=0. To activate the entry for the first SUBCASE, SET ID=1, for the 2nd, set ID=2.
2. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
3. The BCHANGE entry covers Marc's history definitions CONTACT NODE and EXCLUDE.
4. BCHANGE is useful only for saving computer time and is not recommended for general usage.
5. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.

6. If more than one N1-N2-INC range is required for a body, enter N1 as a negative value for all ranges except for the last range for which N1 is entered as a positive value.
7. If more than one N1-N2-INC range is required, all ranges with IBOD1 must come first, followed by all with IBOD2, etc.
8. The EXCLUDE option is obsolete. The reasons why it was added in the past have been alleviated by better improved contact algorithms. If the EXCLUDE option is entered for 3D shapes two N1-N2-INC ranges are normally required to define all 4 nodes of a patch. If the patch is triangular, the last two nodes must be repeated. The following are examples of how data are entered for one element using the exclude option. The first example is for a 4-node patch (nodes 100, 110, 200, 300) and the second is for a 3-node patch (nodes 132, 97, 95, 95).

|         |   |         |     |     |   |      |     |    |  |
|---------|---|---------|-----|-----|---|------|-----|----|--|
| BCHANGE | 1 | EXCLUDE | 1   |     | 1 | -100 | 110 | 10 |  |
|         | 1 | 200     | 300 | 100 |   |      |     |    |  |

|         |   |         |    |   |   |      |     |   |  |
|---------|---|---------|----|---|---|------|-----|---|--|
| BCHANGE | 1 | EXCLUDE | 1  |   | 1 | -132 | 132 | 0 |  |
|         | 1 | -97     | 97 | 0 | 1 | -95  | 95  | 0 |  |
|         | 1 | 95      | 95 | 0 |   |      |     |   |  |

**BCMATL****3D Contact Region by Element Materials**

Defines a 3D contact region by element material. All elements with the specified materials define a contact body used in SOL 600 and SOL 700.

**Format:**

| 1      | 2   | 3   | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|------|-----|-----|-----|-----|-----|----|
| BCMATL | ID  | IM1 | IM2  | IM3 | IM4 | IM5 | IM6 | IM7 |    |
|        | IM8 | IM9 | etc. |     |     |     |     |     |    |

**Example:**

|        |      |     |     |     |  |  |  |  |  |
|--------|------|-----|-----|-----|--|--|--|--|--|
| BCMATL | 1001 | 101 | 201 | 301 |  |  |  |  |  |
|--------|------|-----|-----|-----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control command, BCONTACT=BCMATL is specified. All elements corresponding to the material IDs specified may potentially come into contact. See Remark 1. (Integer > 0) |
| IMi        | Material ID. A minimum of one entry is required. (Integer)  |

**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCPROP entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all material IDs associated with a particular deformable body.
6. BCMATL may not be used to define contact regions made up of composite elements.

**BCMOVE****Movement of Bodies in Contact**

Defines movement of bodies in contact used in SOL 101 and SOL 400.

**Format:**

| 1      | 2       | 3       | 4       | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|---------|---------|---------|------|---|---|---|---|----|
| BCMOVE | ID      | MTYPE   |         |      |   |   |   |   |    |
|        | IDRBOD1 | IDRBOD2 | IDRBOD3 | etc. |   |   |   |   |    |

**Examples:**

|        |    |          |   |   |  |  |  |  |  |
|--------|----|----------|---|---|--|--|--|--|--|
| BCMOVE | 33 | RELEASE  |   |   |  |  |  |  |  |
|        | 1  | 3        | 5 | 7 |  |  |  |  |  |
| BCMOVE | 1  | approach |   |   |  |  |  |  |  |

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

|    |   |
|----|---|
| ID | Identification number referenced by a BCONTACT or BCMOVE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer $\geq 0$ ; Required) |
|----|---|

|       |  |
|-------|--|
| MTYPE | Movement type. (Character; Default = APPROACH) |
|-------|--|

|          |  |
|----------|--|
| APPROACH | All rigid bodies are moved so that they all make contact with deformable bodies. |
|----------|--|

|         |  |
|---------|--|
| RELEASE | The contact condition is released for selected bodies. |
|---------|--|

|          |   |
|----------|---|
| SYNCHRON | All rigid bodies are moved until the first rigid body makes contact with a deformable body. |
|----------|---|

|         |  |
|---------|--|
| IDRBODi | Identification numbers of contact bodies to be released, for option RELEASE only. Points to BCBODY or BCBODY1 Bulk Data entries. |
|---------|--|

**Remarks:**

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCMOVE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCMOVE=ID. Note that if BCMOVE Case Control command exists, it always dominates the selection of BCMOVE Bulk Data entries. ID must be unique (only one BCMOVE per SUBCASE).
2. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
3. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
4. You may release a deformable body from contact with either a deformable or rigid body.

**BCMOVE****Movement of Bodies in Contact in SOL 600**

Defines movement of bodies in contact to be used in SOL 600.

**Format:**

| 1      | 2       | 3       | 4       | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|---------|---------|---------|------|---|---|---|---|----|
| BCMOVE | ID      | MTYPE   | IREL    |      |   |   |   |   |    |
|        | IDRBOD1 | IDRBOD2 | IDRBOD3 | etc. |   |   |   |   |    |

**Examples:**

|        |    |          |    |   |  |  |  |  |  |
|--------|----|----------|----|---|--|--|--|--|--|
| BCMOVE | 33 | RELEASE  | 20 |   |  |  |  |  |  |
|        | 1  | 3        | 5  | 7 |  |  |  |  |  |
| BCMOVE | 1  | approach |    |   |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number referenced by a SUBCASE or STEP Case Control command.<br>See Remark 1. (Integer $\geq 0$ ; Required)                          |
| MTYPE      | Movement type. (Character; Default = APPROACH)  |
| APPROACH   | All rigid bodies are moved so that they all make contact with deformable bodies.  |
| RELEASE    | The contact condition is released for selected bodies.  |
| SYNCHRON   | All rigid bodies are moved until the first rigid body makes contact with a deformable body.   |
| IREL       | Flag to indicate how contact forces are removed, for option RELEASE only. (Integer)   |
| = 0        | Contact forces are immediately removed. (Default)   |
| > 0        | Contact forces are reduced to zero over the number of increments specified in this load period. See NLPARM and TSTEP1 for the number of increments. |
| IDRBODi    | Identification numbers of rigid bodies to be released, for option RELEASE only. Points to BCBODY Bulk Data entries.                                 |

**Remarks:**

1. To place an entry in Marc's phase 0, set ID=0. To activate the entry for the 1st SUBCASE, SET ID=1, for the 2nd, set ID=2. ID must be unique (only one BMOVE per SUBCASE).
2. This entry matches Marc's history definitions RELEASE, APPROACH, and SYNCHRONIZED. Note that Marc's history definition MOTION CHANGE is done in MSC Nastran by describing the enforced motion for the grid which is defined to be the center of rotation of the rigid body, see CGID of the BCBODY entry.

3. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
4. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
5. You may release a deformable body from contact with either a deformable or rigid body.

**BCNURB2**

Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a 2D rigid contact body made up of NURBS used in SOLs 101 and 400 only.

Format:

| 1       | 2                       | 3              | 4              | 5              | 6              | 7      | 8      | 9 | 10 |
|---------|-------------------------|----------------|----------------|----------------|----------------|--------|--------|---|----|
| BCNURB2 | RBID                    | NPTU           | NORU           | NSUB           |                |        |        |   |    |
|         | “GRID”<br>or<br>“COORD” | G1<br>or<br>X1 | G2<br>or<br>Y1 | G3<br>or<br>X2 | G4<br>or<br>Y2 | -etc.- |        |   |    |
|         | “HOMO”                  | Homo1          | Homo2          | Homo3          | Homo4          | -etc.- |        |   |    |
|         | “KNOT”                  | Knot1          | Knot2          | Knot3          | Knot4          | Knot5  | -etc.- |   |    |

Examples:

|         |      |     |     |     |     |     |  |  |  |
|---------|------|-----|-----|-----|-----|-----|--|--|--|
| BCNURB2 | 102  | 4   | 1   | 1   |     |     |  |  |  |
|         | GRID | 237 | 101 | 104 | 235 |     |  |  |  |
|         | HOMO | 0.0 | 0.5 | 0.5 | 1.0 |     |  |  |  |
|         | KNOT | 0.0 | 0.2 | 0.4 | 0.8 | 1.0 |  |  |  |

|         |       |     |         |     |         |         |     |     |  |
|---------|-------|-----|---------|-----|---------|---------|-----|-----|--|
| BCNURB2 | 3001  | -5  | 3       | 50  |         |         |     |     |  |
|         | COORD | -.1 | .14     | -.1 | .04     | -1.4-16 | .04 |     |  |
|         |       | .1  | .04     | .1  | .14     |         |     |     |  |
|         | HOMO  | 1.  | .707107 | 1.  | .707107 | 1.      |     |     |  |
|         | KNOT  | 0.0 | 0.0     | 0.0 | 0.5     | 0.5     | 1.0 | 1.0 |  |
|         |       | 1.0 |         |     |         |         |     |     |  |

**Descriptor**      **Meaning**

**RBID**      Unique identification number referenced by a BCRGSRF or BCBODY1 entry.  
(Integer > 0) See Remark 1.

**NPTU**      Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU must be set to a negative value whose absolute value is the number of xy coordinates. (Integer; no Default)

**NORU**      Order along U direction. (Integer > 0)

**NSUB**      Number of subdivisions. (Integer > 0)

**“GRID”**      Indicate the start of the list of grid numbers. See Remark 2.

**G1, G2, G3, G4,  
etc.**      Grid numbers for each of the NPTU control points. (Integer > 0)

**“COORD”**      Indicate the start of the list of xy coordinates. See Remark 2.

|                              |  |
|------------------------------|--|
| X1, Y1, X2, Y2,<br>etc.      | Alternate method to define control points without using GRID points. There must be<br>abs(NPTU) set of (x,y) entries. (Real) |
| "HOMO"                       | Indicate the start of the list of homogeneous coordinates.   |
| Homo1,Homo2,<br>Homo3, etc.  | Homogeneous coordinates. (Real; 0.0 to 1.0) There must be NPTU entries.  |
| "KNOT"                       | Indicate the start of the list of knot vectors.  |
| Knot1, Knot2,<br>Knot3, etc. | Knot vectors. (Real; 0.0 to 1.0) There must be (NPTU+NORU) entries.  |

**Remarks:**

1. If BCRGSRF entry does not exist, BCNURB2 entry will be referenced by the [BCBODY1](#) entry directly.
2. The list of grid IDs ("GRID") and the list of xy coordinates ("COORD") cannot coexist.

**BCNURBS**

Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a rigid contact body made up of NURBS used in SOLs 101 and 400 only.

**Format:**

| 1       | 2                       | 3              | 4              | 5              | 6              | 7              | 8              | 9      | 10 |
|---------|-------------------------|----------------|----------------|----------------|----------------|----------------|----------------|--------|----|
| BCNURBS | RBID                    | NPTU           | NPTV           | NORU           | NORV           | NSUBU          | NSUBV          |        |    |
|         | “GRID”<br>or<br>“COORD” | G1<br>or<br>X1 | G2<br>or<br>Y1 | G3<br>or<br>Z1 | G4<br>or<br>X2 | G5<br>or<br>Y2 | G6<br>or<br>Z2 | -etc.- |    |
|         | “HOMO”                  | Homo1          | Homo2          | Homo3          | Homo4          | Homo5          | Homo6          | Homo7  |    |
|         |                         | Homo8          | Homo9          | -etc.-         |                |                |                |        |    |
|         | “KNOT”                  | Knot1          | Knot2          | Knot3          | Knot4          | Knot5          | Knot6          | Knot7  |    |
|         |                         | Knot8          | Knot9          | -etc.-         |                |                |                |        |    |
|         | “TRIM”                  | IDtrim1        | IDtrim2        | IDtrim3        | -etc.-         |                |                |        |    |

**Examples:**

|         |      |      |       |     |     |    |    |  |  |
|---------|------|------|-------|-----|-----|----|----|--|--|
| BCNURBS | 48   | 2    | 1     | 2   | 1   | 50 | 50 |  |  |
|         | GRID | 3005 | 102   |     |     |    |    |  |  |
|         | HOMO | 1.   | .3333 |     |     |    |    |  |  |
|         | KNOT | 0.0  | 0.0   | 0.5 | 0.5 | 1. | 1. |  |  |

|         |       |     |      |    |     |     |    |  |  |
|---------|-------|-----|------|----|-----|-----|----|--|--|
| BCNURBS | 63    | -2  | 2    | 2  | 2   | 50  | 50 |  |  |
|         | COORD | .3  | -.6  | 0. | .3  | -.1 | 0. |  |  |
|         | HOMO  | 1.  | 1.   | 1. | 1.  |     |    |  |  |
|         | KNOT  | 0.0 | 0.0  | 1. | 1.  | 0.  | 0. |  |  |
|         |       | 1.  | 1.   |    |     |     |    |  |  |
|         | TRIM  | 511 | 2002 | 87 | 704 |     |    |  |  |

**Descriptor**      **Meaning**

RBID      Unique identification number referenced by a BCRGSRF or BCBODY1 entry.  
(Integer > 0) See Remark 1.

NPTU      Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs, NPTU must be set to a negative value whose absolute value is the number of xyz coordinates. See Remark 2. (Integer > 0; Required)

NPTV      Number of control points in V directions. (Integer > 0; Required)

NORU      Order along U direction. (Integer > 0; Required)

|                                    |  |
|------------------------------------|--|
| NORV                               | Order along V direction. (Integer > 0; Required)   |
| NSUBU                              | Number of subdivisions in U direction. (Integer > 0; Required)   |
| NSUBV                              | Number of subdivisions in V direction. (Integer > 0; Required)   |
| “GRID”                             | Indicate the start of the list of grid numbers. See Remark 3.  |
| G1, G2, G3, etc.                   | Grid point IDs defining control points. (Integer > 0) There must be $\text{NPTU} \times \text{NPTV}$ entries.  |
| “COORD”                            | Indicate the start of the list of xyz coordinates. See Remark 3.   |
| X1, Y1, Z1, X2,<br>Y2, Z2, etc.    | Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) \times \text{NPTV}$ sets of (x,y,z) entries. |
| “HOMO”                             | Indicate the start of the list of homogeneous coordinates.   |
| Homo1, Homo2,<br>Homo3, etc.       | Homogeneous coordinates (0.0 to 1.0). There must be $\text{abs}(\text{NPTU}) \times \text{NPTV}$ entries.<br>(Real)                                      |
| “KNOT”                             | Indicate the start of the list of knot vectors.  |
| Knot1, Knot2,<br>Knot3, etc.       | Knot vectors (0.0 to 1.0). There must be $(\text{abs}(\text{NPTU}) + \text{NORU}) + (\text{NPTV} + \text{NORV})$ entries. (Real)                         |
| “TRIM”                             | Indicate the start of the list of trimming vector IDs.   |
| IDtrimi                            | Trimming vector identification number of a BCTRIM entry. (Integer > 0)   |
| Homot1,<br>Homot2,<br>Homot3, etc. | Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)   |
| Knott1, Knott2,<br>Knott3, etc.    | Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)   |

**Remarks:**

1. If BCRGSRF entry does not exist, BCNURBS entry will be referenced by the BCBODY1 entry directly.
2. Enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points) or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for  $\text{abs}(\text{NPTU})$  points and set NPTU to a negative value.
3. The list of grid IDs (“GRID”) and the list of xyz coordinates (“COORD”) cannot coexist.

**BCONECT**

Defines a Contact Pair

Defines a contact pair used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types of Nastran SOL 400 Getting Started Guide](#)). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103-112 and standard 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types.

Format:

| 1       | 2         | 3      | 4      | 5       | 6        | 7     | 8     | 9     | 10 |
|---------|-----------|--------|--------|---------|----------|-------|-------|-------|----|
| BCONECT | ID        | BCGPID | BCPPID | IDSLAVE | IDMASTER |       |       |       |    |
|         | “SLAVES”  | IDSL1  | IDSL2  | IDSL3   | IDSL4    | IDSL5 | IDSL6 | IDSL7 |    |
|         |           | IDSL8  | IDSL9  | -etc.-  |          |       |       |       |    |
|         | “MASTERS” | IDMA1  | IDMA2  | IDMA3   | IDMA4    | IDMA5 | IDMA6 | IDMA7 |    |
|         |           | IDMA8  | IDMA9  | -etc.-  |          |       |       |       |    |

Examples:

|         |    |     |  |   |      |  |  |  |  |
|---------|----|-----|--|---|------|--|--|--|--|
| BCONECT | 57 | 306 |  | 2 | 1002 |  |  |  |  |
|---------|----|-----|--|---|------|--|--|--|--|

|         |         |     |     |     |  |  |  |  |  |
|---------|---------|-----|-----|-----|--|--|--|--|--|
| BCONECT | 9       |     | 108 |     |  |  |  |  |  |
|         | SLAVES  | 30  | 26  |     |  |  |  |  |  |
|         | MASTERS | 294 | 135 | 528 |  |  |  |  |  |

## Descriptor      Meaning

- |          |   |
|----------|---|
| ID       | Unique identification number referenced by a BCTABL1 entry (Integer $\geq 0$ ). See Remark 1.   |
| BCGPID   | Parameter identification number of a BCONPRG entry (Integer $\geq 0$ or blank). See Remark 2.   |
| BCPPID   | Parameter identification number of a BCONPRP entry (Integer $\geq 0$ or blank). See Remark 2.   |
| IDSLAVE  | Identification number of BCBODY1, BCSURF and BCGRID entry defining the touching body (Integer $\geq 0$ or blank). See Remarks 3. and 4. |
| IDMASTER | Identification number of BCBODY1 and BCSURF entry defining the touched body (Integer $\geq 0$ or blank). See Remarks 5. and 6.          |
| “SLAVES” | Indicates the start of the list of the touching bodies. See Remark 4.   |

|           |  |
|-----------|--|
| IDSLi     | Identification number of BCBODY1 entry defining the touching bodies (Integer $\geq 0$ or blank).                                       |
|           | For SOL 700, leaving IDSL1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES. |
| "MASTERS" | Indicates the start of the list of bodies touched by touching bodies. See Remark 6.  |
| IDMAi     | Identification number of BCBODY1 entry defining touched bodies (Integer $\geq 0$ or blank).  |

**Remarks:**

1. BCONNECT can be selected by the Case Control command BCONTACT=ID to define surface contact if BCTABL1 entry does not exist. See Remarks 2.. and 3.. of BCTABL1 entry.
2. If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
3. A short input to define a single touching body exists if the user provides IDSLAVE. On the other hand, if the user leaves IDSLAVE blank, then "SLAVES" descriptor is required and IDSL1 must be specified. Exceptions are for SOL 700 self-contact, which may use a slave IDSL1 of zero and no "MASTERS" entry.
4. "SLAVES" and IDSLi fields will be ignored if IDSLAVE exists. If IDSLAVE field is blank, then "SLAVES" and IDSLi must be specified. In this case, each IDSLi will be processed separately.
5. A short input to define a single touched body exists if the user provides IDMASTER. On the other hand, if the user leaves IDMASTER blank, then "MASTERS" descriptor is required and IDMA1 must be specified.
6. "MASTERS" and IDMAi fields will be ignored if IDMASTER exists. If IDMASTER field is blank, then "MASTERS" and IDMAi must be specified.
7. The concept of Slave and Master relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
8. If all the BCONPRG that are referenced by a BCONNECT (which is referenced by BCTABL1) have a value of IGLUE  $> 0$ , and this BCONNECT is referenced in the 1st Loadcase (SOL 100\*) or the 1st Step (SOL 400) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

**BCONP**

## Contact Parameters

Defines the parameters for a contact region and its properties for slideline contact in SOL 106 or SOL 129. SOL 400 is the preferred method for contact analysis, see BCBODY1, BCNECT, BCONPRG, BCONPRP, BCPARA and BCTABL1.

Format:

| 1     | 2  | 3     | 4      | 5 | 6    | 7      | 8     | 9   | 10 |
|-------|----|-------|--------|---|------|--------|-------|-----|----|
| BCONP | ID | SLAVE | MASTER |   | SFAC | FRICID | PTYPE | CID |    |

Example:

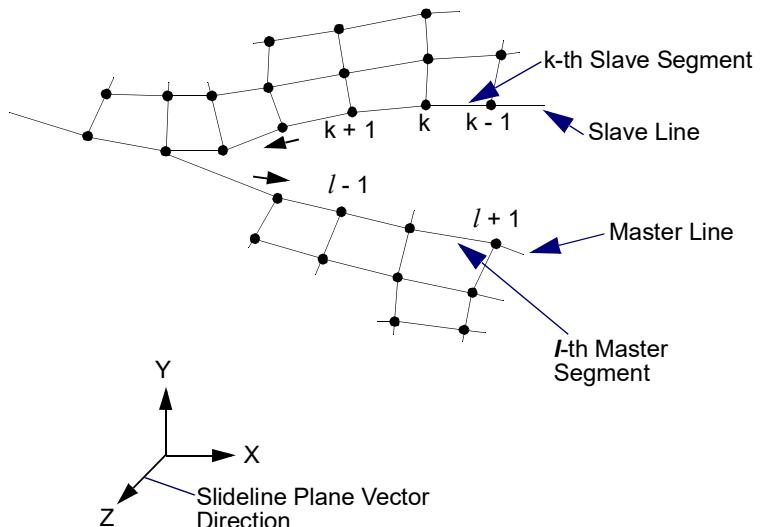
|       |    |    |    |  |     |    |   |  |  |
|-------|----|----|----|--|-----|----|---|--|--|
| BCONP | 95 | 10 | 15 |  | 1.0 | 33 | 1 |  |  |
|-------|----|----|----|--|-----|----|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Contact region identification number. See Remark 1. (Integer > 0)  |
| SLAVE      | Slave region identification number. See Remark 2. (Integer > 0)  |
| MASTER     | Master region identification number. See Remark 3. (Integer > 0)   |
| SFAC       | Stiffness scaling factor. SFAC is used to scale the penalty values automatically calculated by the program. See Remark 4. (Real > 0.0; Default = 1.0)  |
| FRICID     | Contact friction identification number. See Remark 5. (Integer > 0 or blank)   |
| PTYPE      | Penetration type. See Remark 6. (Integer 1 or 2; Default = 1)<br>1           Unsymmetrical (slave penetration only--Default)<br>2           Symmetrical  |
| CID        | Coordinate system identification number to define the slideline plane vector and the slideline plane of contact. See Remark 7. (Integer > 0; Default = 0, which means the basic coordinate system) |

Remarks:

1. ID field must be unique with respect to all other BCONP identification numbers.
2. The referenced SLAVE is the identification number in the BLSEG Bulk Data entry. This is the slave line. The width of each slave segment must also be defined to get proper contact stresses. See the Bulk Data entry, [BWIDTH, 1447](#) for the details of specifying widths.
3. The referenced MASTER is the identification number in the BLSEG Bulk Data entry. This is the master line. For symmetrical penetration, the width of each master segment must also be defined. See the Bulk Data entry, [BWIDTH, 1447](#) for the details of specifying widths.

4. SFAC may be used to scale the penalty values automatically calculated by the program. The program calculates the penalty value as a function of the diagonal stiffness matrix coefficients that are in the contact region. In addition to SFAC, penalty values calculated by the program may be further scaled by the ADPCON parameter (see description of ADPCON parameter for more details). The penalty value is then equal to  $k \cdot SFAC \cdot |ADPCON|$ , where  $k$  is a function of the local stiffness. It should be noted that the value in SFAC applies to only one contact region, whereas the ADPCON parameter applies to all the contact regions in the model.
5. The referenced FRICID is the identification number of the BFRIC Bulk Data entry. The BFRLC defines the frictional properties for the contact region.
6. In an unsymmetrical contact algorithm only slave nodes are checked for penetration into master segments. This may result in master nodes penetrating the slave line. However, the error depends only on the mesh discretization. In symmetric penetration both slave and master nodes are checked for penetration. Thus, no distinction is made between slave and master. Symmetric penetration may be up to thirty percent more expensive than the unsymmetric penetration.
7. In [Figure 9-2](#), the unit vector in the Z-axis of the coordinate system defines the slideline plane vector. The slideline plane vector is normal to the slideline plane. Relative motions outside the slideline plane are ignored, and therefore must be small compared to a typical master segment. For a master segment the direction from master node 1 to master node 2 gives the tangential direction ( $t$ ). The normal direction for a master segment is obtained by the cross product of the slideline plane vector with the unit tangent vector (i.e.,  $n = z \times t$ ). The definition of the coordinate system should be such that the normal direction must point toward the slave region. For symmetric, penetration, the normals of master segments and slave segments must face each other. This is generally accomplished by traversing from master line to slave line in a counterclockwise or clockwise fashion depending on whether the slideline plane vector forms a right-hand or left-hand coordinate system with the slideline plane.



- X-Y plane is the slideline plane. Unit normal in the Z-direction is the slideline plane vector.
- Arrows show positive direction for ordering nodes. Counterclockwise from master line to slave line.
- Slave and master segment normals must face each other.

Figure 9-2 A Typical Finite Element Slideline Contact Region

**BCONPRG****Geometric Contact Parameters of Touching Bodies**

Defines geometric contact parameters used to determine if contact occurs between bodies. For segment-to-segment contact it also defines additional numerical parameters used to apply the constraints. The parameters defined here are referenced by the BCNECT entry. This entry is used in conjunction with the BCONPRP entry.

Geometric Contact Parameters of Touching Bodies in SOLs 101 and 400 for General Contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the Permanently Glued or Tied Contact.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| BCONPRG | BCGPID |      | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 |    |
|         | PARAM4 | VAL4 | PARAM5 | VAL5 | -etc.- |      |        |      |    |

**Example:**

|         |    |  |        |   |       |   |  |  |  |
|---------|----|--|--------|---|-------|---|--|--|--|
| BCONPRG | 90 |  | ICOORD | 1 | IGLUE | 1 |  |  |  |
|---------|----|--|--------|---|-------|---|--|--|--|

**Descriptor**      **Meaning**

BCGPID      Identification number for geometric contact parameters (Integer > 0).

PARAMi      Name of a parameter. Allowable names are given in [Table 9-4](#) (Character).

VALi      Value of a parameter. See [Table 9-4](#) (Real or Integer).

Table 9-4 Geometric Contact Parameters of Touching Bodies in SOLs 101, 103, 105, 107-112, 200 and 400

| Name            | Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)  |
|-----------------|--|
| AUGDIST         | Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real $\geq 0.0$ , see Remark 6. for default)  |
| BIAS            | Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)  |
| CINTERF         | The definition varies depending on OPINTRF. (Real; Default = 0.0)<br>When OPINTRF is set to 0 (default), Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies. For CINTERF < 0.0, gap between bodies.<br>When OPINTRF is set to 1, interference closure magnitude. To clear penetration, set a negative value.<br>When OPINTRF is set to 2, magnitude of translation vector defining in VXINTRF, VYINTRF and VZINTRF.<br>When OPINTRF is set to 3, not used.<br>When OPINTRF is set to 4, not used.                 |
| COPTS,<br>COPTM | Flag to indicate how slave and master surfaces may contact. See Remark 2. (Integer; Default = 1011) COPTS and COPTM apply to all slave and master surfaces on the associated BCONECT.<br><br>COPTS and COPTM are ignored if slave and master in BCONECT are BCSURF or BCGRID entries.  |
| ERROR           | Distance below which a node is considered touching a body. Default = blank, automatic calculation. See BCPARA Bulk Data entry for more details. (Real)   |
| HARDS           | Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, i.e. ISTYP=2 on the BCBDRP entry. The hard-soft ratio can be used by the program if there is a significant difference in the average stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0) This parameter is ignored for the permanently glued contact. |
| FGCFLG          | Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.)<br>0 – no Cohesive (Flexible) Glued Contact<br>1 – activate Cohesive (Flexible) Glued Contact   |
| FGCNST          | Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact   |
| FGCTST          | Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact   |
| FGCNSTR         | ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.  |

|         |  |
|---------|--|
| FGCTSTR | ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.  |
| FGCNSTI | ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.  |
| FGCTSTI | ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.  |
| FGCRCEN | ID of grid on which the resultant contact and moment are based.  |
| ICOORD  | Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)  |
| IGLUE   | Flag to activate glue options (Integer $\geq 0$ ). Default is 0, no glue option. See JGLUE option for controlling a glue contact type in details. This field must be specified for the permanently glued contact. See Remarks 4. and 5. A negative value of IGLUE indicates that this method of glue will be used for the entire Step in SOL 400. See Remark 9.  |
| 1 or -1 | Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body. For the permanently glued contact, this option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified. |
| 2 or -2 | Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.  |
| 3 or -3 | Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body. For the permanently glued contact, this option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified.   |
| 4 or -4 | Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.   |

|         |   |
|---------|---|
| ISEARCH | <p>Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0)</p> <p>0 (Double orders search) the search order is from lower BCBODY1 ID's to higher ones first. Then it searches the opposite order. See Remark 3.</p> <p>1 (Single order search) the searching order is from Slave to Master.</p> <p>2 (Single order search) let the program decide which search order. See Remark 3.</p> <p>Note that ISTYP in <a href="#">BCBDPRP</a> or <a href="#">BCBODY</a> is supported with ISEARCH=0 only.</p> |
| JGLUE   | This option is only relevant if the glue option is invoked (JGLUE > 0). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force (related to FNTOL parameter of BCPARA or BCONPRP entry). Enter 2 to activate breaking glue (related to BGM, BGN, BGSN and BGST parameters of BCONPRP entry). (Integer; Default = 0) This parameter is ignored for the permanently glued contact or step glued contact, see Remark 9.  |
| PENALT  | Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real $\geq 0.0$ ; see Remark 7. for default)   |
| SLIDE   | Delayed slide off distance. This entry should not be made unless ICOORD $\geq 2$ (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default=0.0)  |
| STKSLP  | Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real $\geq 0.0$ ; Default = 0.0) See Remark 8.   |
| TPENALT | Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real $> 0.0$ ) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.   |
| OPINTRF | The method for interference fit. (Integer $\geq 0$ ; Default=0) See Remark 10.<br>0 small interference fit (solved in one increment)<br>1 interference fit varying with time; resolved in the normal direction<br>2 interference fit varying with time in a user specified direction<br>3 interference fit varying with time using scale factor<br>4 automatic interference fit varying with time and node location   |

|         |   |
|---------|---|
| TBINTRF | TABLED1 id used by interference fit. (Integer>0; Required only when OPINTRF > 0)<br>When OPINTRF=1, TABLED1 id giving variation of interference closure with respect to time<br>When OPINTRF=2, TABLED1 id giving variation of magnitude of translation vector with respect to time<br>When OPINTRF=3, TABLED1 id giving variation of magnitude of scale factor with respect to time<br>When OPINTRF=4, TABLED1 id giving variation of magnitude of penetration vector projected node position on the master segment with respect to time |
| CBINTRF | Contact body selection for interference fit. Only required when OPINTRF=2, 3 or 4.<br>(Integer $\geq 0$ ; Default=0)<br>When OPINTRF=2, Contact body ID for which translation vector to be used<br>When OPINTRF=3, Contact body ID for which scale factors to be used.<br>When OPINTRF=4, Contact body ID of which penetrations to be determined.<br>0 slave body<br>1 master body  |
| CDINTRF | Coordinate system ID of VXINTRF, VYINTRF and VZINTRF for interference fit. Only required when OPINTRF=2, 3. (Integer $\geq 0$ ; Default=0)<br>When OPINTRF=2, Coordinate system for direction cosines for the translation vector.<br>When OPINTRF=3, Coordinate system for scale factor vector.   |
| VXINTRF | X component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0)<br>When OPINTRF=2, Direction cosines for the translation vector.<br>When OPINTRF=3, scale factor vector  |
| VYINTRF | Y component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0)<br>When OPINTRF=2, Direction cosines for the translation vector.<br>When OPINTRF=3, scale factor vector.   |
| VZINTRF | Z component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0)<br>When OPINTRF=2, Direction cosines for the translation vector.<br>When OPINTRF=3, scale factor vector.   |
| PTINTRF | Penetration search tolerance for interference fit. Only required when OPINTRF=4.<br>(Real $\geq 0.0$ ; default=error tolerance*100)   |
| XCINTRF | X component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0)<br>Center is described in global coordinate (Coord0).   |
| YCINTRF | Y component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0)<br>Center is described in global coordinate (Coord0).   |
| ZCINTRF | Z component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0)<br>Center is described in global coordinate (Coord0).   |

|         |  |
|---------|--|
| OPINGP  | Initial gap or overlap option. See Remark 1. (Integer $\geq 0$ ; default=0)<br>0 not used<br>1 initial gap or overlap  |
| TOLINGP | Search tolerance of initial gap. See Remark 14.. (Real $\geq 0.0$ ; default= average edge length of all contact segments, required only when OPINGP =1)  |
| TOLING1 | Search tolerance of initial overlap. See Remark 14. (Real $\geq 0.0$ ; default=average edge length of all contact segments, required only when OPINGP =1)  |
| CDINGP  | Contact body to be adjusted. (Integer $\geq 0$ ; default=0, required only when OPINGP =1)<br>0 slave body<br>1 master body   |
| MGINGP  | Gap or overlap magnitude. (Real ; default=0.0, required only when OPINGP =1)<br>> 0 gap<br>=0 Preserve initial clearance distance. See Remark 11.<br>< 0 overlap   |
| SFNPNLT | Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFNPNLT*TPENALT is used in analysis.<br>If this field is left default, value of SFNPNLT entered in BCPARA will be used. A nonblank value entered in SFNPNLT will override the value of SFNPNLT in BCPARA.     |
| SFTPNLT | Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFTPNLT*TPENALT is used in analysis.<br>If this field is left default, value of SFTPNLT entered in BCPARA will be used. A nonblank value entered in SFTPNLT will override the value of SFTPNLT in BCPARA. |

### Remarks:

1. The multipoint constraint equations (MPC equations) internally created from general contact or glued contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH.
2. COPTS and COPTM are packed numbers designating how the surfaces may contact using the formula

$$\text{COPTx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body.

- = 1: the outside will be in the contact description (Default).

B: (flexible bodies): the outside of the shell elements in the body.

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default).
- = 2: only bottom faces will be in the contact description, thickness offset will be included.
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored.

- = 4: only top faces will be in the contact description, thickness offset will be included.
- = 5: only top faces will be in the contact description, shell thickness will be ignored.
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored.

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

Note: for segment-to-segment contact only use B=1 or B=6.

Note: For thermal contact with 2D shell elements, i.e. uniform temperature gradient across thickness, do not use B=2 or B=3.

B: (rigid bodies): the rigid surface.

- = 1: the rigid surface should be in the contact description (Default).

C: (flexible bodies): the edges of the body.

- = 1: only the beam/bar edges are included in the contact description (Default).
- = 10: only the free and hard shell edges are included in the contact description.
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description.

Note that C has no effect if beam-to-beam contact is not switched on. (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

- When ISEARCH=0 (and ISTYP=0 in default on BCBDRP), the search order is from lower BCBODY1 ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY1 ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY1. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2.

CL1= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY1 ID1

CL2= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY1 ID2

The search order is from lower BCBODY1 ID1 to higher BCBODY1 ID2 if CL1≤1.05\*CL2.  
Otherwise, if CL1>1.05\*CL2 the search order is from BCBODY1 ID2 to BCBODY1 ID1.

- For the permanently glued contact, IGLUE = ±2 or ±4 is favorable to pass the GROUNDCHECK. The initial stress free contact is also available to preserve the six rigid body modes with IGLUE = ±1 or ±3.
- A permanently glued contact for small rotation cannot be used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the permanently glued contact for large deformation and rotation (IGLUE <0) or general contact (IGLUE > 0) with glued option must be performed.
- By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

AUGDIST=0.05 L

7. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$\text{PENALT} = \frac{0.5(S_i + S_j)}{L}$$

The body stiffness ( $S_i$  and  $S_j$ ), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length ( $L$ ) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and is given by

$$\text{PENALT} = \frac{1000S_i}{L}$$

8. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ( $\mu \cdot F_{\text{normal}}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
9. A negative value of IGLUE enforces the gluing of the pair of bodies in the BCONECT that reference this BCONPRG is based upon the geometric conditions at the beginning of the step and will not change over this step. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.
10. For interference fit contact, the following options are available.  
CINTERF, OPINTRF, TBINTRF, CBINTRF, CDINTRF, VXINTRF, VYINTRF, VZINTRF, PTINTRF, XCINTRF, YCINTRF and ZCINTRF
11. For setting the initial gap or overlap, the following options are available. OPINGP, TOLINGP, CDINGP and MGINGP.

When OPINGP is 1 in means that there it will try to preserve a distance between the two bodies often called the clearance distance.

When MGINGP is zero, the clearance is not changed from the distance between the original coordinate positions of the nodes. This is call the distance D0

When MGINGP is  $> 0$  it means the distance between the bodies can shrink (i.e. the bodies can get closer together), but when the distance reaches  $D0 - MGINGP$ . The clearance condition is satisfied and this clearance distance ( $D0 - MGINGP$ ) is preserved.

When MGINGP is  $< 0$  in means that it considers the bodies to be overlapping, so it first tries to separate them by a distance  $= |MGINGP|$ ; then it imposes the clearance condition such that the clearance distance is  $D0 + |MGINGP|$

12. Please note that
  - Cohesive Contact is supported for only for segment-to-segment contact.
  - All limitations of segment-to-segment contact is applicable for cohesive contact feature also.
  - Breaking Glue feature is not supported with Cohesive contact.
13. When the initial stress-free(ICOORD=1), Node-to-Segment updates model geometry to close the gap, nodes are projected onto the contact body; but Segment-to-Segment treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
14. TOLINGP and TOLING1 follows the rules below.
  - a. When both of TOLINGP and TOLING1 are not defined, the default value of average edge length of all contact segments will be used for both.
  - b. When TOLINGP is only defined, TOLINGP value will be used for both of TOLINGP and TOLING1.
  - c. When TOLING1 is only defined, TOLING1 value will be used for TOLING1 only, and the default value of average edge length of all contact segments will be used for TOLINGP.
  - d. When both of TOLINGP and TOLING1 are defined, each values are used for each case

**BCONPRG-700**

## Geometric Contact Parameters of Touching Bodies in SOL 700

Defines geometric contact parameters of touching bodies used in SOL 700 only.

Format:

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 | 11 |
|---------|--------|------|--------|------|--------|------|--------|------|----|----|
| BCONPRP | PID    |      | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 |    |    |
|         | PARAM4 | VAL4 | PARAM5 | VAL5 | -etc.- |      |        |      |    |    |

Example:

|         |    |  |       |   |  |  |  |  |  |  |
|---------|----|--|-------|---|--|--|--|--|--|--|
| BCONPRP | 90 |  | IGLUE | 1 |  |  |  |  |  |  |
|---------|----|--|-------|---|--|--|--|--|--|--|

Descriptor      Meaning

PID              Parameter identification number (Integer > 0).

PARAMi          Name of a parameter. Allowable names are given in [Table 9-5](#) (Character).

VALi              Value of a parameter. See [Table 9-5](#) (Real or Integer).

Table 9-5 Geometric Contact Parameters of Touching Bodies in SOL 700

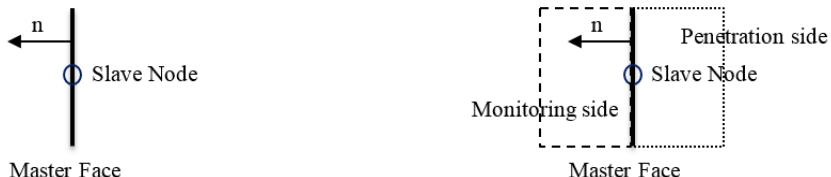
| Name   | Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)  |
|--------|--|
| IGLUE  | <p>Flag to activate glue option (Integer = 0 or 1). Default is 0, no glue option. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, tied contact methods (METHOD=TIEDxxx) can assign glue contact types as well.</p> <ul style="list-style-type: none"> <li>0. No glue contact.</li> <li>1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.</li> </ul> |
| JGLUE  | <p>This option is only relevant if the glue option is invoked (IGLUE &gt; 0). See METHOD=TIEDxxx. (Integer ≥ 0; Default = 0) The following options are available:</p> <ul style="list-style-type: none"> <li>0 Slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.</li> </ul>  |
| METHOD | <p>Character, Influences the contact type used. See Remark 18. (Character, Default = FULL)</p> <p>Options are:</p> <ul style="list-style-type: none"> <li>FULL: Regular Contact</li> <li>AIRBAG: Single Surface Contact</li> <li>SS1WAY: Surface To Surface One Way</li> <li>SS2WAY: Surface To Surface Two Way</li> <li>RB1WAY: Rigid Body One Way To Rigid Body</li> <li>RB2WAY: Rigid Body Two Way To Rigid Body</li> <li>RNRB: Rigid Nodes To Rigid Body</li> <li>TIEDNS: Tied Nodes to Surface</li> <li>TIEDNSO: Tied Nodes to Surface with Offset</li> </ul>   |
| ADAPT  | <p>Character, influences the contact type used. (Character, Default = NO)</p> <p>Options are NO or YES.</p> <p>When ADAPT=YES, the BCBODY1 entries IDMAi must be defined as:</p> <ul style="list-style-type: none"> <li>behav=DEFORM</li> <li>bsid references a BCPROP</li> </ul>  |
| THICK  | Shell thickness scale factor. (Real > 0.0; Default = same as the value in DYPARAM,CONTACT,THICK)   |

|         |   |
|---------|---|
| THICKOF | Artificial contact thickness offset. (Real > 0.0; Default = 0.0)  |
| PENV    | Overwrites the default maximum penetration distance factor. (Real > 0.0;Default = 1.E20)  |
| MAXPAR  | Maximum parametric coordinate in segment. (Real > 0.0; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.   |
| SOFT    | <p>Soft constraint option: (Integer <math>\geq</math> 0; Default = 1)</p> <p>1 Soft constraint formulation (same as original Dytran method).</p> <p>In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method is more suited for contact between two materials where the elastic moduli vary greatly.</p>  |
| IGNORE  | <p>Ignore initial penetrations. (Integer, Default = 1)</p> <p>0 Take default from PARAM,DYCONIGNORE*,&lt;value&gt; (Default)</p> <p>1 Allow initial penetrations to exist by tracking it</p> <p>2 Move nodes to eliminate initial penetrations</p>  |
| AUTO    | <p>Activation of automatic contact (Character, Default = Yes.)</p> <p>Options are:</p> <p>YES Automatic Contacts Activated</p> <p>NO Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.</p>   |
| SIDE    | <p>Defines which side will be the monitoring side of a master face. The opposite side of the master face will be the penetration side. See Remark 1. (Characters; default=BOTH)</p> <p>BOTH: The side from which a slave node approaches the master face will become the monitoring side.</p> <p>TOP: The monitoring side will always be on the side of the master face that its normal is pointing at.</p> <p>BOTTOM: The monitoring side is always on the opposite side of the master face that its normal is pointing at</p> |

|         |  |
|---------|--|
| WEIGHT  | <p>The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH)</p> <p>BOTH: <math>M_{scale} = (M_{slave} M_{master}) / (M_{slave} + M_{master})</math></p> <p>SLAVE: <math>M_{scale} = M_{slave}</math></p> <p>MASTER: <math>M_{scale} = M_{master}</math></p> <p>NONE: <math>M_{scale} = 1.0</math></p>   |
| MONDIS  | <p>Defines the fixed part of the monitoring distance. When the normal distance of a slave node to a master face becomes smaller than the monitoring distance the slave node will tag itself to the master face. The side from which the slave node is moving towards the master face becomes the monitoring region. (Character; default=FACTOR)</p> <p>FACTOR: The monitoring distance is equal to a factor times a characteristic length of the master faces. The factor is specified in MONDISV.</p> <p>DISTANCE: The monitoring distance is specified in MONDISV.</p> |
| MONDISV | Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)  |
| INITMON | Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real > 0.0; default=MONDIS)  |

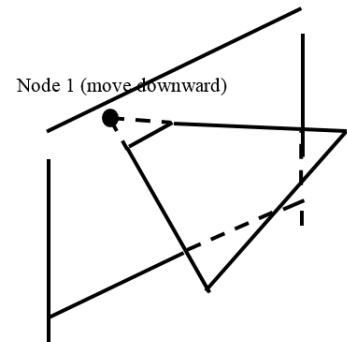
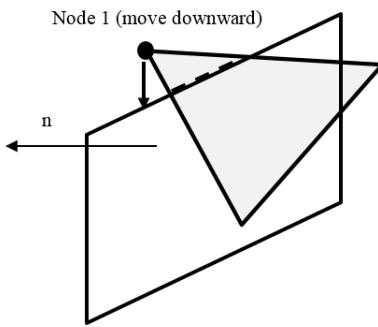
### Remarks:

- When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SLAVE body and BCELIPS is only available for MASTER body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
- The options TOP/BOTTOM are useful in the following cases:
  - When a slave node initially is located on the master face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the master surface is defined:  
SIDE=TOP case:



- When hooking of slave nodes on the wrong side of a master face might occur. This often is the case when the master face is at the edge of a shell element structure:

SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



### 3. Recommended usage of WEIGHT:

| SLAVE      | MASTER     | WEIGHT |
|------------|------------|--------|
| Deformable | Deformable | BOTH   |
| Deformable | Rigid      | SLAVE  |
| Rigid      | Deformable | MASTER |
| Rigid      | Rigid      | NONE   |

Default setting when BCELIPS is used in SLAVE or MASTER:

| SLAVE       | MASTER      | WEIGHT |
|-------------|-------------|--------|
| Non-BCELIPS | Non-BCELIPS | NONE   |
|             | BCELIPS     |        |
|             |             |        |
|             |             |        |

**BCONPRP****Physical Contact Parameters of Touching Bodies in SOLs 101 and 400**

Defines physical contact parameters of touching bodies used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCONECT entry. This entry is used in conjunction with the BCONPRG entry.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| BCONPRP | PID    |      | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 |    |
|         | PARAM4 | VAL4 | PARAM5 | VAL5 | -etc.- |      |        |      |    |

**Example:**

|         |    |  |      |     |  |  |  |  |  |
|---------|----|--|------|-----|--|--|--|--|--|
| BCONPRP | 90 |  | FRIC | 0.3 |  |  |  |  |  |
|---------|----|--|------|-----|--|--|--|--|--|

**Describer      Meaning**

PID      Parameter identification number (Integer > 0).

PARAMi      Name of a parameter. Allowabe names are given in [Table 9-6](#) (Character).

VALi      Value of a parameter. See [Table 9-6](#) (Real or Integer).

Table 9-6 Physical Contact Parameters of Touching Bodies in SOLs 101 and 400

| Name   | Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)  |
|--------|--|
| BGM    | Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)  |
| BGN    | Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)  |
| BGSN   | Maximum normal stress for breaking glue. See Remark 1. (Real; Default = 0.0)   |
| BGST   | Maximum tangential stress for breaking glue. See Remark 1. (Real; Default = 0.0)   |
| BNC    | Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0.)        |
| BNL    | Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0) |
| DQNEAR | Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. See Remark 2. (Real; Default = 0.0; which means near contact does not occur)   |
| EMISS  | Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs. temperature or a TABL3D entry specifying the emissivity vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)   |
| FNTOL  | Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. Its default value is dependent on the IBSEP value. (Real)   |
| FRIC   | Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq 0.0$ or Integer $> 0$ ; Default = 0.0)   |
| FRLIM  | Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction stress will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$ , with $\mu$ the friction coefficient and $\sigma_n$ the contact normal stress. (Real; Default = 1.0E20)  |

|       |  |
|-------|--|
| HBL   | Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs. temperature or a TABL3D entry specifying the convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)  |
| HCT   | Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs. temperature or a TABL3D entry specifying the contact heat transfer coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.e., when IGLUE > 0.) |
| HCV   | Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs. temperature or a TABL3D entry specifying the near field convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)   |
| HGLUE | Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. Ignored in a pure structural analysis. See Remarks 3, 4, and 5. (0 ≤ Integer ≤ 1; Default = 0)   |
| HNC   | Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0).  |
| HNL   | Heat transfer coefficient for nonlinear convective heat flow for near field behavior. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs. temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)   |

#### Remarks:

- Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BGSN = 0.0 the first term is ignored. Similarly, the second term is ignored if BGST = 0.0. For SOL 400, parameters BGSN, BGST, BGM and BGN are only required if JGLUE = 2. If both BGSN and BGST are equal to 0.0, JGLUE will be set to 0 internally. If Step Glue is used, then Breaking Glue is inactive in this step. If Permanent Glue is active, then Breaking Glue is inactive.

2. For near contact the following convective heat flow law describes the heat exchange per unit area between the body areas (“near” contact is when bodies are not touching each other – but are close enough for convection and radiation.):

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_B^4) +$$

$$\left[ HCT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A^4 - T_B^4)$$

where the last term is only activated when HBL ≠ 0,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

3. For hard contact (i.e., two bodies are actually touching each other – heat transfer mode between the bodies is conduction), with HGLUE=1:
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
  - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures i.e. Gibbs Phenomenon or the temperature oscillations at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

4. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

5. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D). Table IDs of tables used on the BCBDPRP entry and the BCONPRP entry must be unique.
6. Parameters BGM, BGN, BGSN, BGST and DQNEAR are not supported in segment to segment contact analysis.

**BCONPRP-700****Physical Contact Parameters of Touching Bodies in SOL 700**

Defines physical contact parameters of touching bodies used in SOL 700 only.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8          | 9    | 10 |
|---------|--------|------|--------|------|--------|------|------------|------|----|
| BCONPRP | PID    |      | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM<br>3 | VAL3 |    |
|         | PARAM4 | VAL4 | PARAM5 | VAL5 | -etc.- |      |            |      |    |

**Example:**

|         |    |  |      |     |  |  |  |  |  |
|---------|----|--|------|-----|--|--|--|--|--|
| BCONPRP | 90 |  | FRIC | 0.3 |  |  |  |  |  |
|---------|----|--|------|-----|--|--|--|--|--|

**Descriptor      Meaning**

PID      Parameter identification number (Integer > 0).

PARAMi      Name of a parameter. Allowable names are given in [Table 9-7](#) (Character).

VALi      Value of a parameter. See [Table 9-7](#) (Real or Integer).

**Table 9-7      Physical Contact Parameters of Touching Bodies in SOL 700**

|      |   |
|------|---|
| Name | Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)   |
| FRIC | Static friction coefficient. When it is set to -1.0, different static friction coefficients on properties can be assigned using BCPROP1. (Real $\geq$ -1.0; Default=0.0 for SOL700) |
| FK   | Kinetic coefficient of friction. (Real $>$ 0.0; Default = 0.0)  |
| EXP  | Exponential decay coefficient. (Real $>$ 0.0; Default = 0.0)  |
| FACT | Scale factor for the contact forces. (Real $>$ 0.0; Default = 0.1)  |

|         |  |
|---------|--|
| TSTART  | Time at which the contact is activated. (Real > 0.0; Default = 0.0)  |
| TEND    | Time at which the contact is deactivated. (Real > 0.0; Default = 1.e20)  |
| IADJ    | <p>Adjacent material treatment option for solid elements. (Integer <math>\geq 0</math>; Default =1)</p> <p>0 Solid element faces are included only for free boundaries.</p> <p>1 Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.</p>   |
| DAMPING | <p>Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a slave node with respect to a master face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES).</p> <p>YES damping is activated<br/>     NO damping is not activated.</p> |

**BCONUDS****Contact Body User Defined Service**

Allows the user to provide contact routines for use with enhanced SOL 400 contact analysis.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| BCONUDS | BID    | BTYPE  | GROUP  | UNAME  |        |        |        |        |    |
|         | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5 | IDATA6 | IDATA7 |    |
|         |        | IDATA8 | IDATA9 | ...    | ...    | IDATAn |        |        |    |
|         | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5 | RDATA6 | RDATA7 |    |
|         |        | RDATA8 | RDATA9 | ...    | ...    | RDATAn |        |        |    |
|         | “CHAR” | CDATA1 | CDATA2 | ...    | ...    | CDATAn |        |        |    |

**Example:**

*In FMS Section of MSC Nastran input stream:*

```
CONNECT SERVICE CONTACT 'SCA.MDSolver.Util.Ums'
```

*In Bulk Data:*

|         |    |        |         |       |  |  |  |  |  |
|---------|----|--------|---------|-------|--|--|--|--|--|
| BCONUDS | 17 | BCBODY | CONTACT | UFRIC |  |  |  |  |  |
|---------|----|--------|---------|-------|--|--|--|--|--|

|         |      |        |         |        |  |  |  |  |  |
|---------|------|--------|---------|--------|--|--|--|--|--|
| BCONUDS | 17   | BCBODY | CONTACT | SEPFOR |  |  |  |  |  |
|         | REAL | .00134 | 1.467+4 | .03    |  |  |  |  |  |

|         |      |        |         |        |  |  |  |  |  |
|---------|------|--------|---------|--------|--|--|--|--|--|
| BCONUDS | 17   | BCBODY | CONTACT | MOTION |  |  |  |  |  |
|         | REAL | .00134 | 1.467+4 | .03    |  |  |  |  |  |
|         | INT  | 8      | 3       |        |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Contact body identification number defined by BCBODY or BCBODY1. (Integer > 0; Required)                                  |
| BTYPE      | The name of the contact entry. BCBODY/BCBODY1 (Character; no Default)   |
| GROUP      | The group name used for the FMS Section CONNECT SERVICE statement. (Character; no Default)                                |
| UNAME      | User subroutine name associated with the entry. See Remark 6. and 7. (Character)  |
| “INT”      | Keyword indicating that the following data is integer. (Character)  |
| IDATAi     | Additional user supplied Integer data not already existing on the specified contact property entry. (Integer, no Default) |
| “REAL”     | Keyword indicating that the following data is real. (Character)   |

| Descriptor | Meaning   |
|------------|---|
| RDATAi     | Additional user supplied Real data not already existing on the specified contact property entry. (Real; no Defaults)          |
| "CHAR"     | Keyword indicating that the following data is Character. (Character)  |
| CDATAi     | Additional user supplied Character data not already existing on the specified contact property entry. (Character; no Default) |

**Remarks:**

1. This entry triggers the call to a user contact subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The BID must match an existing BID.
4. A CDATAi entry cannot be the Character "REAL", "INT", or "CHAR".
5. Certain user subroutines may require integer or real data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. The following user subroutines are currently available for user convenience. See the User Defined Services manual for details.

| Type   | Uname    | Purpose  |
|--------|----------|--|
| BCBODY | motion   | Define velocity of rigid surfaces  |
| BCBODY | ufric    | Define friction coefficients   |
| BCBODY | sepfor   | Define separation forces   |
| BCBODY | sepstr   | Define separations stresses  |
| BCBODY | ubsqueal | Define friction coefficient and effective stiffness in brake squeal analysis |

**BCONUDS****Contact Body User Subroutines in SOL 600**

Allows the user to provide contact routines for use with SOL 600 contact analysis.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| BCONUDS | BID    | BTYPE  |        | UNAME  |        |        |        |        |    |
|         | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5 | IDATA6 | IDATA7 |    |
|         |        | IDATA8 | IDATA9 | ...    | ...    | IDATAn |        |        |    |
|         | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5 | RDATA6 | RDATA7 |    |
|         |        | RDATA8 | RDATA9 | ...    | ...    | RDATAn |        |        |    |

**Example:**

|         |    |        |  |      |  |  |  |  |  |
|---------|----|--------|--|------|--|--|--|--|--|
| BCONUDS | 17 | BCBODY |  | RTN1 |  |  |  |  |  |
|---------|----|--------|--|------|--|--|--|--|--|

|         |      |        |         |      |  |  |  |  |  |
|---------|------|--------|---------|------|--|--|--|--|--|
| BCONUDS | 17   | BCBODY |         | RTN2 |  |  |  |  |  |
|         | REAL | .00134 | 1.467+4 | .03  |  |  |  |  |  |

|         |      |        |         |      |  |  |  |  |  |
|---------|------|--------|---------|------|--|--|--|--|--|
| BCONUDS | 17   | BCBODY |         | RTN3 |  |  |  |  |  |
|         | REAL | .00134 | 1.467+4 | .03  |  |  |  |  |  |
|         | INT  | 8      | 3       |      |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| BID        | Contact body identification number defined by BCBODY. (Integer > 0; Required)  |
| BTYPE      | The name of the contact entry. BCBODY (Character; no Default)  |
| UNAME      | The Marc user subroutine name. See Remarks 2. and 5. (Character; Required)   |
| “INT”      | Keyword indicating that the following data is integer. (Character)   |
| IDATAi     | Additional user supplied Integer data not already existing on the specified MAT entry. See Remark 6. (Integer; no Default) |
| “REAL”     | Keyword indicating that the following data is real. (Character)  |
| RDATAi     | Additional user supplied Real data. See Remark 6. (Real; no Defaults)  |

**Remarks:**

1. The BID must match an existing BID.
2. Certain user subroutines may require integer or real data input as specified in the UserSubroutine interface document.
3. The following user subroutines are currently available for user convenience. See the User Subroutine interface document for details.

| UNAME      | Purpose  |
|------------|--|
| MOTION     | Define velocity of rigid surfaces  |
| UFRIC      | Define friction coefficients   |
| DIGEOM     | Define rigid surfaces  |
| SEPFOR     | Define separation forces   |
| SEPPFORBBC | Define separation forces for beam-beam contact   |
| SEPSTR     | Define separations stresses  |
| UFRICBBC   | Define the friction coefficients for beam-to-beam contact with node-to-segment contact.<br><br>This user subroutine is not used for segment-to-segment contact; use UFRIC instead. |
| UHTNRC     | Define thermal near contact film coefficients  |
| UMDCOE     | Define variable mass and diffusion coefficients and sink pressures on a free surface   |
| UMDCON     | Define variable mass diffusion coefficients on surfaces in contact with other surfaces   |
| UMDNRC     | Define mass diffusion coefficients of surfaces in contact with other surfaces  |
| UNDRST     | Define normal stresses for elements in contact   |

4. The BCONUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character\*16 name in common block /userch/
5. UNAME identifies the user subroutine name to be called.
6. IDATA and RDATA are not normally required. They are available should additional arguments beyond those described in Marc Vol D be required. Normally only the first line is entered.

**BCPARA**

## Contact Parameters -SOLs 101 and 400

Defines contact parameters used in SOL 101 and SOL 400. This entry is not available in SOL 700.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9 | 10 |
|--------|--------|--------|--------|--------|--------|--------|--------|---|----|
| BCPARA | ID     | Param1 | Value1 | Param2 | Value2 | Param3 | Value3 |   |    |
|        | Param4 | Value4 | Param5 | Value5 | etc.   |        |        |   |    |

**Example:**

|        |       |     |      |     |  |  |  |  |
|--------|-------|-----|------|-----|--|--|--|--|
| BCPARA | ERROR | 0.1 | BIAS | 0.5 |  |  |  |  |
|--------|-------|-----|------|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Subcase to which the defined parameters belong. If ID is zero, the parameters belong to all subcases. (Integer $\geq 0$ ; no Default) |
| Param(i)   | Name of a parameter. Allowable names are given in <a href="#">Table 9-7</a> . (Character)   |
| Value(i)   | Value of the parameter. See <a href="#">Table 9-7</a> . (Real or Integer)   |

Table 9-8 Contact Parameters -SOLs 101 and 400

| Name    | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)   |  |
|---------|---|--|
| ERROR   | Distance below which a node is considered touching a body. (Real; Default = blank). Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest value resulting from: Either dividing the smallest nonzero element dimension (plates or solids) in the contact body by 20. Or dividing the thinnest shell thickness in the contact body by 4. This value is then used for all contact pairs. |  |
| BIAS    | Contact tolerance bias factor. (0.0 ≤ Real ≤ 1.0.);<br>Default = 0.9 for IGLUE=0, if field left blank.<br>Default = 0.0 for IGLUE <>0. Note that when IGLUE<>0, BIAS can only be given by the BCTABLE or BCONPRG.<br>Default = 0.0 for BEHAVE=SYMM on BCBODY, if field left blank or 0.0.   |  |
| FGCFLG  | Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.)  |  |
|         | 0   | No Cohesive (Flexible) Glued Contact       |
|         | 1   | Activate Cohesive (Flexible) Glued Contact |
| FGCNST  | Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact  |  |
| FGCTST  | Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact  |  |
| FGCNSTR | ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.   |  |
| FGCTSTR | ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.   |  |
| FGCNSTI | ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.   |  |
| FGCTSTI | ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.   |  |
| FGCRCEN | ID of grid on which the resultant contact and moment are based.   |  |
| FNTOL   | Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank). See Remark 10.   |  |
| MAXSEP  | Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)   |  |

| Name   | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)                           |  |
|--------|---|--|
| ICSEP  | Flag to control separation. Not used for segment-to-segment contact. (Integer $\geq 0$ ; Default = 0)                     |  |
|        | 0   | The node separates and an iteration occurs if the force on the node is greater than the separation force.  |
|        | 1   | If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment. |
|        | 2   | If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.  |
|        | 3   | Both 1 and 2 are in effect.  |
| IBSEP  | Flag for separation based on stresses or forces. (Integer $\geq 0$ ; Default = 0)   |  |
|        | 0   | Separation based on forces.  |
|        | 1   | Separation based on absolute stresses (force/area)   |
|        | 2   | Separation based on absolute stress (extrapolating integration point stresses)   |
|        | 3   | Relative nodal stress (force/area)   |
|        | 4   | Separation based on relative stress (extrapolating integration point stresses)   |
|        | Segment to segment contact ignores IBSEP and will use 2 internally. See Remarks <a href="#">6</a> and <a href="#">8</a> . |  |
| RVCNST | For the bilinear friction model, RVCNST is the slip threshold. (Real $\geq 0.0$ ; Default = 0.0).                         |  |
|        | Default=0.0 means that the actual value is automatically calculated.  |  |
| FTYPE  | Friction type. See Remark <a href="#">5</a> . (Integer)   |  |
|        | 0   | No friction. (Default)   |
|        | 6   | Bilinear Coulomb friction.   |
|        | 7   | Bilinear Shear friction.   |
| BEAMB  | Beam-Beam contact flag. See Remark <a href="#">7</a> . (Integer 0 or 1)   |  |
|        | 0   | No beam-beam contact. (Default for SOLs 101/400)   |
|        | 1   | Activate beam-beam contact options. (Default for SOL 700)  |

| Name                              | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)   |   |
|-----------------------------------|---|---|
| NLGLUE<br>(SOLs 101 and 400 only) | If all slave's for the BCTABLE or BCONPRG corresponding to the first loadcase (first subcase and first step) contain IGLUE >0, permanent glued contact with small rotation condition will be used for all SLAVE entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified. If IGLUE < 0 exists, permanent glued contact for large deformation and large rotation is activated. (Integer; Default = 0) |   |
| NODSEP                            | Skip separation check in contact for any grid already touched and separated.  NODSEP  times within the current increment. (Integer; Default = 5)  |   |
|                                   | > 0   | Keep grid contact status  |
|                                   | < 0   | Separate grid from contact  |
| METHOD                            | Flag to select Contact methods. (Character)   |   |
|                                   | NODESURF  | Node to segment contact. (Default)  |
|                                   | SEGTOSSEG   | Segment to segment contact.   |
| AUGMENT                           | Augmentation method used in a segment-to-segment contact analysis. (Integer)  |   |
|                                   | 0   | No augmentation (Default)   |
|                                   | 1   | Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements |
|                                   | 2   | Augmentation based on a constant Lagrange multiplier field  |
|                                   | 3   | Augmentation based on a (bi)linear Lagrange multiplier field  |
| PENALT                            | Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real > 0.0; see BCTABLE or BCONPRG entries for default)  |   |
| AUGDIST                           | Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real > 0; see BCTABLE entry for default)   |   |
| SLDLMT                            | Maximum allowed sliding distance, beyond it the contact segments are redefined, for segment to segment contact analysis with large deformation. (Real $\geq$ 0.0; Default = 0.0) See remark 9.  |   |
| SEGSYM                            | Specify symmetric or non-symmetric friction matrix in segment to segment contact analysis. (Integer 0 = symmetric matrix or 1 = non-symmetric matrix; Default = 0)  |   |
| ERRBAS                            | Error computation option. (Integer 0 = compute error globally or 1 = calculate error based on each pair of slave-master; Default = 0)   |   |
| DDULMT                            | Maximum value of DDU in a Newton-Raphson cycle. (Real $\geq$ 0.0; Default = 0.0, no limitation)   |   |
| TAUGMNT                           | Augmentation for the sticking part of friction in a segment-to-segment contact analysis. (Integer 0 = no augmentation or 1 = use augmentation; Default = 0)   |   |

| Name    | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)   |
|---------|---|
| TPENALT | Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real > 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.  |
| STKSLP  | Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real $\geq$ 0.0; Default = 0.0) See Remark 11..  |
| THKOFF  | Ignore thickness from the tolerance used by ISEARCH=2 in node-to-surface contact or from the characteristic length (for PENALT and AUGDIST) in segment-to-segment contact. (Integer 0 = do not ignore thickness or 1 = remove thickness; Default = 0)   |
| LINCNT  | Flag for Linear Contact (under infinitesimal assumption, small sliding with small deformation and rotation). Supports both node-to-segment and segment-to-segment method. See Remark 14..<br>0 - (Default) general contact<br>1 - activate Linear Contact, the contact force is distributed based upon undeformed geometry.<br>-1 - force Linear Contact with large displacement (not recommended). |
| SFNPNLT | Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0)<br>SFNPNLT*PENALT is used in analysis.<br>A nonblank value of SFNPNLT entered in BCONPRG will override it.   |
| SFTPNLT | Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0)<br>SFTPNLT*TPENALT is used in analysis.<br>A nonblank value of SFTPNLT entered in BCONPRG will override it.  |
| SEGANGL | Minimum angle between segment normal vectors allowing the segments to come into contact; used by segment-to-segment contact only. (Real>90.0 <180.0; Default=120.0).  |

**Remarks:**

1. The ID of BCPARA should be left blank or a value of zero. Blank is treated as zero.
2. Only one BCPARA entry should be made. Multiple BCPARA entries with ID=0 cause a Fatal Error.
3. All SOLs only support ID=0, this entry is applied to all the subcases, except the corresponding ones are assigned in BCTABLE or BCONPRG.
4. Refer to the Case Control command [BCONTACT \(Case\)](#), to see the lists of parameters in BCPARA that are not supported by SOLs 101 and 400.

5. In SOLs 101/400, FTYPE only supports options 0, 6, and 7. Friction with segment-to-segment contact is a pre-release capability in the 2012 release. In the 2012.2 version and later, it is fully supported.
6. Shell elements (CQUAD4 CQUAD8 CTRIA3 CTRIA6) in 3D Contact only supports IBSEP = 0, 1 and 3. If IBSEP = 2 or 4 in this case, it will automatically be set to 1 or 3, respectively. In SOLs 101/400, LINQUAD is not supported and it is set to -1 in order that the mid-side nodes are considered in Contact. Quadratic shell elements consider mid-side nodes only in 2D Contact.
  - a. If IBSEP is set to 1, then for bodies consisting of mid-side node elements, IBSEP will automatically be set to 2. This allows for using IBSEP=1 for elements without mid-side nodes and IBSEP=2 for elements with mid-side nodes in one analysis.
  - b. If IBSEP is set to 3, then for bodies consisting of mid-side nodes elements, IBSEP will automatically be set to 4. This allows for using IBSEP=3 for elements without mid-side nodes and IBSEP=4 for elements with mid-side nodes in one analysis.

This rule is always applied to the solid elements with mid-side nodes. However, the mid-side nodes of CQUAD8 and CTRIA6 shell elements are ignored in node-to-segment 3D contact with separation because there is no normal stress in shell elements, and the contact normal forces for these elements do not give a good estimation of the nodal contact stress. Hence it is not recommended to use CQUAD8 and CTRIA6 shell elements in a node-to-segment contact analysis with separation.

7. For SOLs 101/400 if BEAMB=1, Bulk Data entry BCBMRAD must be entered. Those entries are not required in the beam contact of segment-to-segment contact.
8. Parameters MAXSEP, ICSEP, IBSEP, RVCNST and BEAMB are not supported in segment to segment contact analysis.
9. The default of SLDELM parameter is 5 times the default error tolerance. See the description of ERROR parameter for the definition of default error tolerance.
10. IBSEP and FNTOL are related to each other. The Default value of FNTOL is dependent on IBSEP value.
  - a. IBSEP=0 : Separation is based on forces. A contact node will separate if the contact pulling force exceeds FNTOL. If FNTOL is zero it will be taken as the absolute value of the largest component in the residual force vector.
  - b. IBSEP=1 : Separation is based on nominal contact stresses (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL. If FNTOL is zero, it will be 0.1 times of the maximum nominal contact pushing stress of any node of any contact body.
  - c. IBSEP=2 : Separation is based on nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL. If FNTOL is zero it will be 0.1 times of the maximum nodal pushing stress of any node of any contact body
  - d. IBSEP=3 : Separation is based on relative nominal contact stress (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL times of maximum nominal contact pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1
  - e. IBSEP=4 : Separation is based on relative nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL times of maximum nodal pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1

11. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ( $\mu \cdot F_{NORMAL}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
12. The parameters of cohesive contact defined in BCPARA will be applied to all the contact pairs unless:
  - The contact pair has its own definition for cohesive contact in BCONPRG; or
  - Its IGLUE=0, i.e., general touching contact.Cohesive contact is only applied to glued contact, but not touching contact.
13. If Modules are present then this entry may only be specified in the main Bulk Data section.
14. LINCNT does not support RIGID contact body with velocity control.

**BCPARA****Contact Parameters in SOL 600**

Defines contact parameters used in SOL 600.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9 | 10 |
|--------|--------|--------|--------|--------|--------|--------|--------|---|----|
| BCPARA | ID     | Param1 | Value1 | Param2 | Value2 | Param3 | Value3 |   |    |
|        | Param4 | Value4 | Param5 | Value5 | etc    |        |        |   |    |

**Example:**

|        |  |         |   |      |     |  |  |  |  |
|--------|--|---------|---|------|-----|--|--|--|--|
| BCPARA |  | NBODIES | 4 | BIAS | 0.5 |  |  |  |  |
|--------|--|---------|---|------|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | ID is not used and should be set to zero. Only one BCPARA should be entered and it applies to all subcases. (Integer) |
| Param(i)   | Name of a parameter. Allowable names are given in <a href="#">Table 9-7</a> . (Character)                             |
| Value(i)   | Value of the parameter. See the <a href="#">Table 9-7</a> . (Real or Integer)   |

| Name          | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)  |   |
|---------------|--|---|
| NBODIES (2,1) | Number of contact bodies defined in the analysis. (Integer $\geq 0$ or blank)  |   |
| MAXENT (2,2)  | Maximum number of entities created for any contact body. (Integer $> 0$ or blank; default is max element number or 1.5 times the number of nodes whichever is smaller)   |   |
| MAXNOD (2,3)  | Maximum number of nodes that lie on the periphery of any deformable contact body. (Integer $> 0$ or blank; default is the number of nodes)   |   |
| ERROR (3,2)   | Distance below which a node is considered touching a body. Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest nonzero element dimension divided by 20 or the shell thickness divided by 4. (Real; Default = blank) |   |
| BIAS (3,6)    | Contact tolerance bias factor. (Real, $0 \leq \text{BIAS} \leq 1$ , Default = 0.9 if field left blank or not entered in the file. To obtain a near zero value, enter 1.0E-16)  |   |
| ISPLIT (2,7)  | Flag for increment splitting procedure. (Integer $\geq 0$ ; Default = 3 for statics and 0 for dynamics)  |   |
|               | 0  | Uses increment splitting procedures for the fixed time step procedures. |

| Name         | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated) |  |
|--------------|---|--|
|              | 1   | Suppresses splitting for the fixed time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods                                       |
|              | 2   | Suppresses splitting for adaptive time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods.                                       |
|              | 3   | Uses contact procedure which does not require increment splitting (3 is not available for dynamics). If a run does not converge due to an excessive number of “iterative penetration checking” messages, ISPLIT=2 may help, however the time steps may need to be set to smaller values. |
| FNTOL (3,5)  |   | Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank)  |
| MAXSEP (2,6) |   | Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)  |
| ICHECK (2,8) |   | Flag for interference kinematic check and bounding box check. (Integer > 0)  |
|              | 1   | Activates interference kinematic check.  |
|              | 2   | Suppress bounding box checking.  |
|              | 3   | No reset of NCYCLE to zero.  |
|              | 4   | Check for separation only when solution has converged, for analytical surfaces only.   |
| ICSEP (2,9)  |   | Flag to control separation. (Integer $\geq 0$ ; Default = 0)   |
|              | 0   | The node separates and an iteration occurs if the force on the node is greater than the separation force.  |
|              | 1   | If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment.   |
|              | 2   | If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.  |
|              | 3   | Both 1 and 2 are in effect.  |
| IBSEP (2,12) |   | Flag for separation based on stresses or forces. (Integer $\geq 0$ ; Default = 0)  |
|              | 0   | Separation based on forces.  |

| Name          | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated) |   |
|---------------|---|---|
|               | 1   | Separation based on absolute stresses (force/area).   |
|               | 2   | Separation based on absolute stress (extrapolating integration point stresses).   |
|               | 3   | Relative nodal stress (force/area).   |
|               | 4   | Separation based on relative stress (extrapolating integration point stresses).   |
|               |   | Only option 2 and 4 can be used with mid-side node elements where the mid-side nodes contact (LINQUAD=-1).  |
| ISHELL (2,10) |   | Parameter governing normal direction and thickness contribution of shells. (Integer $\geq 0$ ; Default = 0)   |
|               | 0   | Check node contact with top and bottom surface.   |
|               | 1   | Nodes only come into contact with bottom layer.   |
|               | 2   | Nodes only come into contact with bottom layer and shell thickness is ignored.  |
|               | -1  | Nodes only come into contact with top layer.  |
|               | -2  | Nodes only come into contact with top layer and shell thickness is ignored.   |
| IPRINT (2,11) |   | Flag to reduce print out of surface definition. (Integer $\geq 0$ ; Default = 0)  |
|               | 0   | Full print out.   |
|               | 1   | Reduced print out.  |
| RVCNST (3,1)  |   | Relative sliding velocity between bodies below which sticking is simulated. If FTYPE=5, then the value of RVCNST is the stick-slip transition region. |
| FTYPE (2,4)   |   | Friction type. (Integer)  |
|               | 0   | No friction. (Default)  |
|               | 1   | Shear friction.   |
|               | 2   | Coulomb Friction.   |
|               | 3   | Shear friction for rolling.   |
|               | 4   | Coulomb friction for rolling.   |
|               | 5   | Stick-slip Coulomb friction.  |
|               | 6   | Bilinear Coulomb friction. (Default if friction is present and FTYPE is not entered.)   |
|               | 7   | Bilinear Shear friction.  |
| FKIND (2,5)   |   | Friction kind. (Integer 0 or 1)   |

| Name              | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)  |   |
|-------------------|--|---|
|                   | 0  | Friction based on nodal stress.   |
|                   | 1  | Default if friction is present and FKIND is not entered. Friction based on nodal force.   |
| BEAMB<br>(2,13)   | Beam-Beam contact flag. (Integer 0 or 1)   |   |
|                   | 0  | No beam-beam contact. (Default)   |
|                   | 1  | Activate beam-beam contact options.   |
| FSSMULT<br>(3,7)  | Stick-slip friction coefficient multiplier. Applicable only to stick-slip friction. (The friction coefficient is multiplied by this value for the sticking condition.) (Real $\geq 0$ ; Required; Default = 1.05)  |   |
| FSSTOL<br>(3,8)   | Stick-slip friction force tolerance. Applicable only to stick-slip friction. (Real; Default = 0.05)  |   |
| LINQUAD (2,14)    | Higher order element contact flag (Integer; Default = 1).  |   |
|                   | 1  | The outer boundary of a contact body is described by the corner nodes only and mid-side nodes can't come into contact.  |
|                   | -1   | The outer boundary is described by a quadratic field and both corner and mid-side nodes are considered in contact. If this flag is set to -1 and IBSEP is blank, IBSEP will be re-set to 2. This option is only available with Marc 2003 and subsequent releases. |
| INITCON<br>(2,16) | If INITCON is set, tying relations (MPC's) for surfaces initially in contact will be saved. This option maybe used to model dissimilar meshes. See CONTINUE=101+ on the SOL 600 entry to use these items in the same MSC Nastran execution. The following options are available:   |   |
|                   | 1  | MPC's in Marc format are saved for initial contact (if any) in file jid.marc.t01.   |
|                   | 3  | MPC's for each increment are saved in Marc format in file jid.marc.conmpc_xxxx where xxxx is the increment number.  |
|                   | 4  | MPC's for each increment are saved in Nastran MPC format in file jidd.marc.conmpc_xxxx where xxxx is the increment number.  |
|                   | When initcon=1, the job will stop at the end of increment zero. When initcon=3 initcon=4, the job will run to completion and the information in jid.marc.conmpc_0000 for increment zero will usually not be useful, but the information for increments one and above will contain the proper contact tying relations or mpc's. |   |
| NVSURF            | Rigid contact surface ID (BCBODY ID) for which the next 4 approach velocity values apply. Leave this entry and the next 4 out if not applicable. This entry and the next 4 may be repeated as many times as necessary to define all rigid contact surfaces with approach velocity values. (Integer; Default = 0)               |   |

| Name          | Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)  |  |
|---------------|--|--|
| VELAPP1 (6,1) | Approach velocity in direction 1 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)  |  |
| VELAPP2 (6,2) | Approach velocity in direction 2 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)  |  |
| VELAPP3 (6,3) | Approach velocity in direction 3 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)  |  |
| VELAPAN (6,3) | Approach angular velocity about local axis through center of rotation of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,1 (Real; Default = 0.0)  |  |
| NODSEP        | If NODSEP is positive, ignore separation of any node that has touched and separated NODSEP times (corresponds to feature, 31xx where xx=nodsep). If NODSEP is negative, NODSEP is the number of separations of a particular node allowed. If more separations of that node actually occur, the node is assumed to be separated (corresponds to feature, -31xx where xx=nodsep). (Integer between 0 and 99; Default if not entered is 2, corresponds to feature,3102) |  |
| METHOD        | Flag to select Contact Methods (Character)   |  |
|               | NODES URF  | Regular 3D contact (Default: node to surface contact)  |
|               | SEGSMA LL  | Segment to segment contact with small sliding  |
| AUGMENT       | Augmentation method used in a segment to segment contact analysis. (Integer)   |  |
|               | 0  | No augmentation (Default)  |
|               | 1  | Augmentation based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements. |
|               | 2  | Augmentation based on a constant Lagrange multiplier field.  |
|               | 3  | Augmentation based on a bilinear Lagrange multiplier field.  |
| PENALT        | Augmented Lagrange penalty factor used by the segment to segment contact algorithm only. (Real > 0; set BCBODY for Default)  |  |
| AUGDIST       | Penetration distance beyond which an augmentation will be applied. Used by the segment to segment contact algorithm only. (Real > 0.0; set BCBODY for Default)   |  |

**Remarks:**

- (i,j) refers to data block i and field j of the CONTACT model definition option in Marc.
- Field 2 of the primary line should be left blank or a value of zero should be entered.
- Only one BCPARA entry should be made. If multiple entries are made, the last will be used.

4. For FTYPE, SOLs 400 and 600 differ as follows. For SOL 400, if friction is entered but FTYPE is blank, friction is ignored. For SOL 600, if friction is entered but FTYPE is blank COULOMB friction is used. (FTYPE is reset internally to 2). In addition SOL 400 can only use friction types 0, 6 and 7.

**BCPATCH**

Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400

Defines a rigid contact body made up of quadrilateral patches used in SOLs 101 and 400 only.

**Format:**

| 1       | 2      | 3  | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|---------|--------|----|----|----|----|---|---|---|----|
| BCPATCH | RBID   |    |    |    |    |   |   |   |    |
|         | IDP    | G1 | G2 | G3 | G4 |   |   |   |    |
|         | IDP    | G1 | G2 | G3 | G4 |   |   |   |    |
|         | -etc.- |    |    |    |    |   |   |   |    |

**Example:**

|         |    |     |     |     |     |  |  |  |  |
|---------|----|-----|-----|-----|-----|--|--|--|--|
| BCPATCH | 12 |     |     |     |     |  |  |  |  |
|         | 1  | 101 | 102 | 103 | 104 |  |  |  |  |

**Descriptor      Meaning**

RBID      Unique identification number referenced by a BCRGSRF or BCBODY1 entry (Integer > 0). See Remark 1.

IDP      ID of the patch (Integer number 1 through highest value).

G1, G2, G3,  
G4      Grid numbers for each of the 4 nodes of the patch.

**Remarks:**

1. If BCRGSRF entry does not exist, BCPATCH entry will be referenced by the BCBODY1 entry directly.

**BCPFLG**

Defines a beam section segment flag

Defines a beam section branch and segment for segment to segment beam contact used in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued Contact.

Format:

| 1      | 2   | 3       | 4           | 5       | 6       | 7       | 8       | 9 | 10 |
|--------|-----|---------|-------------|---------|---------|---------|---------|---|----|
| BCPFLG | PID | IBRNCH1 | IOUTIN<br>1 | IBRNCH2 | IOUTIN2 | IBRNCHi | IOUTINi |   |    |

Example:

|        |   |   |   |   |    |  |  |  |  |
|--------|---|---|---|---|----|--|--|--|--|
| BCPLAG | 2 | 1 | 2 | 3 | 13 |  |  |  |  |
|--------|---|---|---|---|----|--|--|--|--|

**Descriptor      Meaning**

PID      Beam property/section identification number of PBEAML/PBTRL. (Integer > 0; Required)

IBRNCHi      Branch selection flag for segment generation. Only required for thin wall sections. See Remark 1, 2, 3 and 4. (Integer; default=0).

0 select all branches for thin wall sections

IOUTINi      Beam segment location. See Remark 4, 5, 6 and 7. (Integer; default=0; up to three unique integers may be placed in the field with no embedded blanks.)

0 all segments

1 location 1 of thin wall section or only outer of solid or tube sections

2 location 2 of thin wall section or only inner of tube section

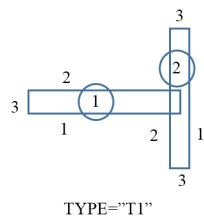
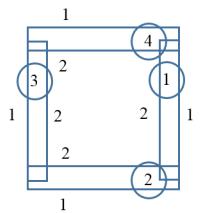
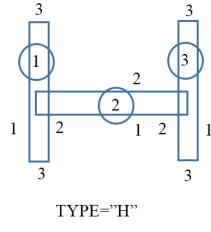
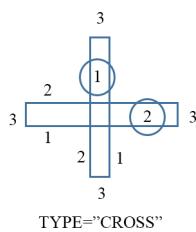
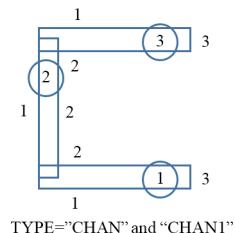
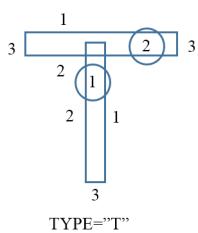
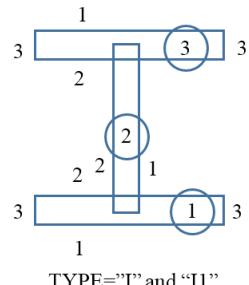
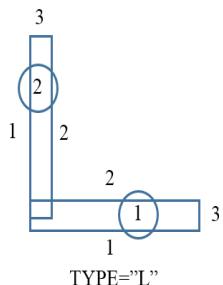
3 location 3 of thin wall section (thickness direction of branch)

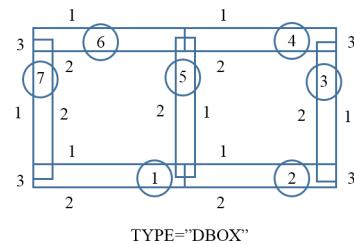
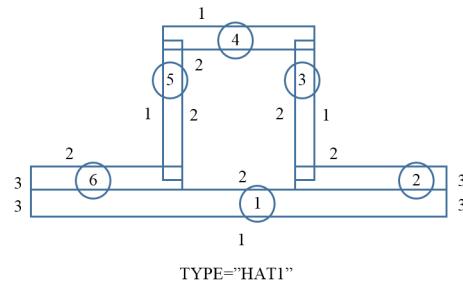
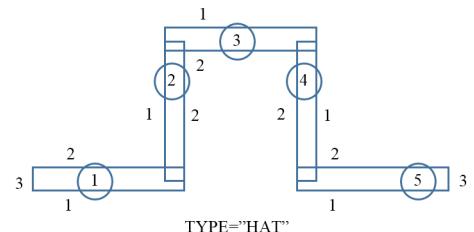
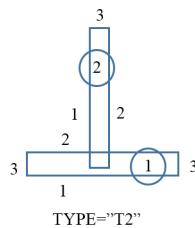
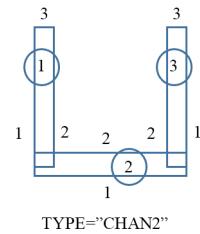
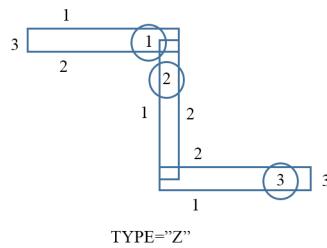
4 segment not including thickness

Remarks:

1. If the card is not defined but the beam property is included in segment to segment contact, all segments will be generated and used. This card is only useful when a user needs to define the specific beam segments for segment to segment contact. It will help the performance of calculation because only selected segments will be considered in the contact calculation.
2. IBRNCHi fields are only required for thin wall beam sections. Solid beam sections such as rectangular, rod, etc or tube sections require only IOUTIN1.
3. Each branch of thin wall beam section has four segments.
4. Segment locations of thin wall beam sections are listed below. TYPE is defined by PBEAML/PBTRL. The number circled is the branch number of each shape and the number without circle is the segment number which is used in IOUTINi. Segment 1 and 2 of each branch are the segments in the branch direction and segment 3's are the segment in the thickness direction of the branch.

Defines a beam section segment flag





5. If there is not thickness, one segment will be generated in the branch.
6. "3" cannot be used alone and "4" must be used only with "1" or "2". IOUTINi must be one of "0", "1", "2", "12", "13", "14", "23", "24" and "123".
7. "14" and "24" generate the identical segment since the thickness is ignored but segment normal direction will be different.

**BCPROP****3D Contact Region by Element Properties**

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOL 101 and SOLs 400 and 700 only.

**Format:**

| 1      | 2   | 3   | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|------|-----|-----|-----|-----|-----|----|
| BCPROP | ID  | IP1 | IP2  | IP3 | IP4 | IP5 | IP6 | IP7 |    |
|        | IP8 | IP9 | etc. |     |     |     |     |     |    |

**Example:**

|        |   |     |     |     |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|
| BCPROP | 1 | 101 | 201 | 301 |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|

**Alternate Format:**

|        |     |     |      |      |     |     |     |     |  |
|--------|-----|-----|------|------|-----|-----|-----|-----|--|
| BCPROP | ID  | IP1 | THRU | IP2  | IP3 | IP4 | IP5 | IP6 |  |
|        | IP7 | IP8 | IP9  | etc. |     |     |     |     |  |

**Alternate Example:**

|        |   |     |      |     |     |      |     |     |  |
|--------|---|-----|------|-----|-----|------|-----|-----|--|
| BCPROP | 1 | 101 | THRU | 102 | 105 | THRU | 109 | 110 |  |
|--------|---|-----|------|-----|-----|------|-----|-----|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0) |
| IPi        | Property ID. A minimum of one entry is required. (Integer; no Default)  |

**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU. The THRU keyword may appear in fields 4 thru 8.
7. BCBOX and BCMATL are not available for SOL 101 or SOL 400

**BCPROP****3D Contact Region by Element Properties in SOL 600**

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOL 600.

**Format:**

| 1      | 2   | 3   | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|------|-----|-----|-----|-----|-----|----|
| BCPROP | ID  | IP1 | IP2  | IP3 | IP4 | IP5 | IP6 | IP7 |    |
|        | IP8 | IP9 | etc. |     |     |     |     |     |    |

**Example:**

|        |   |     |     |     |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|
| BCPROP | 1 | 101 | 201 | 301 |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|

**Alternate Format:**

|        |    |     |      |     |  |  |  |  |  |
|--------|----|-----|------|-----|--|--|--|--|--|
| BCPROP | ID | IP1 | THRU | IP2 |  |  |  |  |  |
|--------|----|-----|------|-----|--|--|--|--|--|

**Example for Alternate Format:**

|        |    |     |      |     |  |  |  |  |  |
|--------|----|-----|------|-----|--|--|--|--|--|
| BCPROP | 25 | 101 | THRU | 102 |  |  |  |  |  |
|--------|----|-----|------|-----|--|--|--|--|--|

**Descriptor      Meaning**

- |     |  |
|-----|--|
| ID  | Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry or if the Case Control BCCONTACT = BCPROP is specified. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0) |
| IPi | Property ID. A minimum of one entry is required. (Integer; no Default)   |

**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU.

**BCRIGID**

Defines a Rigid Contact Body in SOLs 101 and 400

Defines a rigid contact body used in SOLs 101 and 400 only.

Format: (SOLs 101 and 400 only)

| 1       | 2        | 3      | 4       | 5     | 6       | 7       | 8       | 9      | 10 |
|---------|----------|--------|---------|-------|---------|---------|---------|--------|----|
| BCRIGID | BCRGID   | CGID   | CONTROL |       |         |         |         |        |    |
|         | NLOAD    | ANGVEL | DCOS1   | DCOS2 | DCOS3   | VELRB1  | VELRB2  | VELRB3 |    |
|         | "APPROV" | A      | N1      | N2    | N3      | V1      | V2      | V3     |    |
|         | "GROW"   | GF1    | GF2     | GF3   | TAB-GF1 | TAB-GF2 | TAB-GF3 |        |    |

Example:

|         |      |    |    |    |    |    |    |    |  |
|---------|------|----|----|----|----|----|----|----|--|
| BCRIGID | 1001 | 1  | 1  |    |    |    |    |    |  |
|         | 0    | 0. | 0. | 0. | 1. | 1. | 1. | 1. |  |

**Describer**      **Meaning**

BCRGID      Unique identification number referenced by a BCBODY1 entry. (Integer > 0)

CGID      Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied. (Integer > 0) See Remark 1.

CONTROL      Indicates the type of control for the body.

Integer:

= -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRB<sub>i</sub> in the following line.

= 0 for velocity control. (default)

= positive number for "load control". The positive number is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)

NLOAD      Enter a positive number if "load controlled" and rotations are allowed. (Integer) The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for translation about x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively). Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation.

ANGVEL      Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)

|          |   |
|----------|---|
| DCOSi    | Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.   |
| VELRBi   | Translational velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID field. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0) |
| “APPROV” | The entries of this continuation line are for approaching velocity to establish initial contact.  |
| A        | Angular velocity about local axis through center of rotation. (Real, Default = 0.0)   |
| Ni       | Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the CGID field. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)   |
| Vi       | V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)   |
| “GROW”   | The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2 (growth vs time) or TABL3D (growth vs one or more variables).  |
| GFi      | Components of growth factor of rigid body in the coordinate system of the CGID field. (Real, Default = 1.0)   |
| TAB-GFi  | Table IDs for growth factor of rigid body in the coordinate system of the CGID field. (Integer > 0 or blank, Default is blank)  |

#### Remarks:

1. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID. If CGID is not specified, basic coordinate will be used.
2. All continuation lines may be omitted if not required.

**BCRGSRF****Rigid Contact Surface List in SOLs 101 and 400**

Defines a list of rigid contact surfaces used in SOLs 101 and 400 only.

**Format:**

| 1       | 2     | 3      | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|--------|-------|-------|-------|-------|-------|-------|----|
| BCRGSRF | BSID  | RBID1  | RBID2 | RBID3 | RBID4 | RBID5 | RBID6 | RBID7 |    |
|         | RBID9 | -etc.- |       |       |       |       |       |       |    |

**Example:**

|         |      |    |   |  |  |  |  |  |  |
|---------|------|----|---|--|--|--|--|--|--|
| BCRGSRF | 1008 | 35 | 2 |  |  |  |  |  |  |
|---------|------|----|---|--|--|--|--|--|--|

**Descriptor      Meaning**

BSID      Unique identification number referenced by a BCBODY1 entry. (Integer > 0) See Remark 1.

RBIDi      Identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry. (Integer > 0)

**Remarks:**

1. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry directly.

**BCSCAP**

Defines a cap and number of segments

Defines whether a cap is added to a beam section and/or defines the number of segments used to model rod and tube sections for segment to segment beam contact in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued contact

**Format:**

| 1      | 2   | 3      | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|--------|------|---|---|---|---|---|----|
| BCSCAP | EID | IESCAP | NSEG |   |   |   |   |   |    |

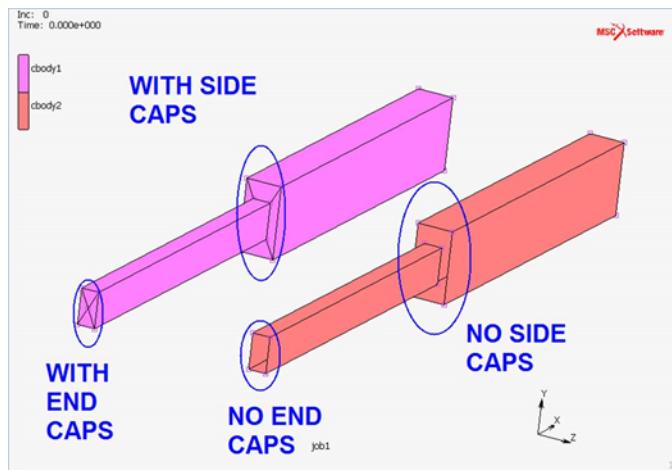
**Example:**

| 1      | 2 | 3 | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---|---|----|---|---|---|---|---|----|
| BCSCAP | 2 | 1 | 20 |   |   |   |   |   |    |

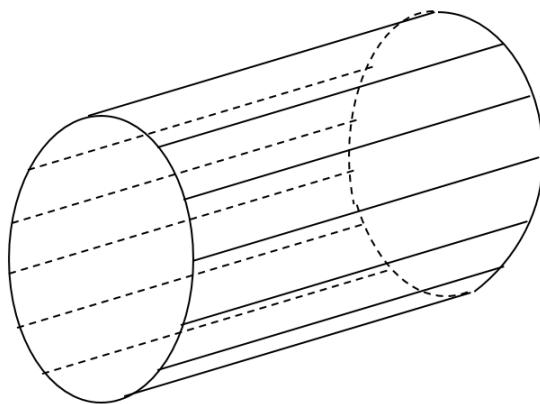
| Descriptor | Meaning   |
|------------|---|
| EID        | Beam element identification number. (Integer > 0; Required)   |
| IESCAP     | End and side cap flag (Integer $\geq 0$ ; default=0).<br>0 no end or side cap<br>1 end cap only<br>2 side cap only<br>3 averaged side cap only<br>4 end cap and side cap<br>5 end cap and averaged side cap |
| NSEG       | Number of segments for circular sections TYPE=ROD, TUBE or TUBE2 beam sections defined on PBARL or PBEAML. See remarks 2. (Integer $\geq 3$ ; default=32)   |

**Remarks:**

1. If the card is not defined but the beam is included in segment to segment contact, all caps will be ignored. This card is only useful when a user needs to define the cap of beams for segment to segment contact.



2. The segments for circular cross sections will be generated as the figure below.



**BCSEG**

## Contact Segment Defined Using Grids

Grids which are part of an element to be used in contact analyses in SOL 700 only.

Format:

| 1     | 2  | 3     | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|-------|----|-------|----|----|----|----|---|---|----|
| BCSEG | ID | IBODY | G1 | G2 | G3 | G4 |   |   |    |

Example:

| 1     | 2   | 3    | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|-------|-----|------|----|----|----|----|---|---|----|
| BCSEG | 100 | 1005 | 11 | 12 | 13 | 14 |   |   |    |

| Descriptor  | Meaning  |
|-------------|--|
| ID          | Unique identification number for this BCSEG entry. (Integer > 0; Required)   |
| IBODY       | Identification number of a surface that is called out on the 5 <sup>th</sup> field of a BCBODY entry. (Integer > 0; Required)  |
| G1,G2,G3,G4 | GRID point identification numbers of an element on this surface. For quad plates and quad surfaces of solids, enter four grid id's. For triangular plates or triangular faces of solids, leave G4 blank. (Integer > 0; Required) |

Remarks:

1. This entry is used as shown in the example below:

BCBODY, 201,,1005

BCSEG,1,1005,11,12,13,14

BCSEG,2,1005,21,22,23,24

BCSEG,3,1005,31,32,33,34

(In the above 11-14, 21-24 and 31-34 are GRID ID's)

**BCSURF****Contact entry by element faces**

Define a contact region by element faces for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400. This has three forms based the FORM (the 6th) field. "FACE" (default) is to define contact face by element face IDs, "GRID" is to define the same by grid point ids, and "RIGID" is to define a rigid face.

**Format 1: FORM="FACE" (default)**

| 1      | 2     | 3       | 4       | 5 | 6     | 7       | 8       | 9 | 10 |
|--------|-------|---------|---------|---|-------|---------|---------|---|----|
| BCSURF | BID   | BPID    | DIM     |   | FORM  | INCTHK  | EDGCNT  |   |    |
|        | ESID1 | FACEID1 | IDTYPE1 |   | ESID2 | FACEID2 | IDTYPE2 |   |    |
|        | ESID3 | FACEID3 | IDTYPE3 |   | ...   |         |         |   |    |
|        | ...   |         |         |   |       |         |         |   |    |

**Format 2: FORM="GRID"**

| 1      | 2    | 3    | 4   | 5   | 6      | 7      | 8      | 9   | 10 |
|--------|------|------|-----|-----|--------|--------|--------|-----|----|
| BCSURF | BID  | BPID | DIM |     | "GRID" | INCTHK | EDGCNT |     |    |
|        | EID1 | G11  | G12 | G13 | EID2   | G21    | G22    | G23 |    |
|        | EID3 | G31  | G32 | G33 | ...    |        |        |     |    |
|        | ...  |      |     |     |        |        |        |     |    |

**Format 3: FORM="RIGID"**

| 1      | 2    | 3      | 4   | 5 | 6       | 7 | 8 | 9 | 10 |
|--------|------|--------|-----|---|---------|---|---|---|----|
| BCSURF | BID  | BPID   | DIM |   | "RIGID" |   |   |   |    |
|        | BSID | BCRGID |     |   |         |   |   |   |    |

Example:

**FACE form (default)**

|        |    |    |    |  |   |    |     |  |  |
|--------|----|----|----|--|---|----|-----|--|--|
| BCSURF | 2  | 3  | 3D |  |   | NO |     |  |  |
|        | 12 | S2 |    |  | 4 | S4 | SET |  |  |

**GRID form**

|        |    |   |    |   |      |     |    |    |  |
|--------|----|---|----|---|------|-----|----|----|--|
| BCSURF | 3  | 3 | 3D |   | GRID | YES |    |    |  |
|        | 11 | 1 | 3  | 6 | 16   | 31  | 14 | 15 |  |

**RIGID form**

|        |     |     |    |  |       |  |  |  |  |
|--------|-----|-----|----|--|-------|--|--|--|--|
| BCSURF | 4   |     | 3D |  | RIGID |  |  |  |  |
|        | 101 | 201 |    |  |       |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Unique contact face identification number referenced by BCNECT (Integer > 0; Required). See remark <a href="#">1</a> .  |
| BPID       | Parameter identification number of a BCBDRP entry. (Integer > 0 or blank).  |
| DIM        | Dimension of body. (Character; Default= “3D”), “3D” or “2D”.  |
| FORM       | Select a type of the entry. (Character; Default =”FACE”).<br>FACE define the contact surface by element face IDs<br>GRID define the contact surface by grid points on an element face<br>RIGID define the contact surface by rigid surface  |
| INCTHK     | Option to include shell thickness offset. (Character; Default =”YES”), “YES” or “NO”.   |
| EDGCNT     | Option for edge contact. (Integer, default=1), below three values can be specified.<br>1: only the beam/bar edges are included in the contact description (Default).<br>10: only the free and hard shell edges are included in the contact description.<br>11: both the beam/bar edges and the free and hard shell edges are included in the contact description. |
| ESIDI      | Element id or set id. (Integer, no default). If IDTYPEi is “SET”, it is a set id defined by a SET3, and the SET3 must be element type. If IDTYPEi is “ELEM” (default), it is an element id.   |
| FACEIDI    | Element face ID. (Character, blank can be used to express BOTH for shell elements and the whole bodies of beam or bar elements). See remarks <a href="#">6</a> , <a href="#">7</a> , <a href="#">8</a> . and <a href="#">9</a> .  |
| IDTYPEi    | Type of ESIDI. (Character, “ELEM”).<br>ELEM ESIDI is an element id.<br>SET ESIDI is a SET3 id.  |
| EIDI       | Element id. (Integer > 0)   |
| GIDIj      | Three corner grid point ids of the element face. (Integer, no default). All the three fields of for an EIDI are blanks or zeros mean BOTH for shell elements and the whole bodies of bar or beam elements. See remark <a href="#">2</a> .   |
| BSID       | Identification number of a BCRGSRF, BCPATCH, BCBZIER, BCNURB2, or BCNURBS. (Integer > 0).   |
| BCRGID     | Identification number of a BCRIGID entry. (Integer > 0).  |

**Remarks:**

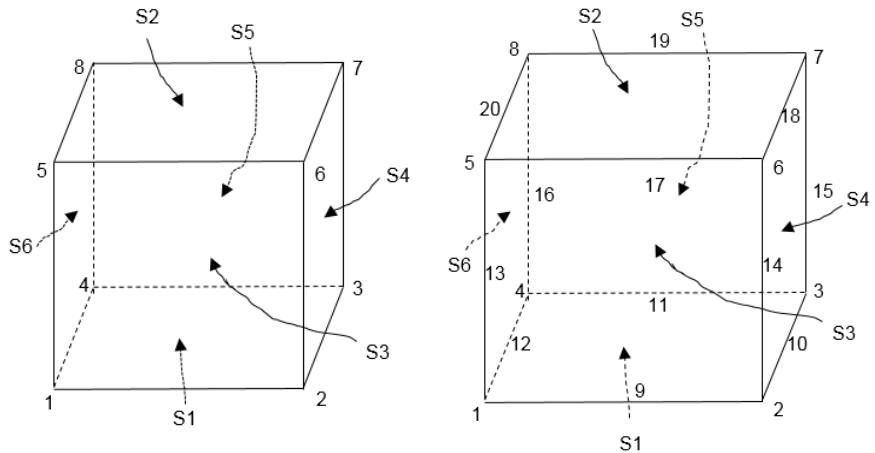
1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.

2. BCSURF entries are able to coexist with BCBODY1 entries, however a BCSURF can only be used to construct a contact pair with another BCSURF, itself or a BCGRID in a BCONECT.
3. BCSURF cannot be specified in the continuations of a BCONECT.
4. If a BCSURF or BCGRID is referenced by a BCONECT, COPTS and COPTM in the BCONPRG referenced by the BCONECT are ignored.
5. For GRID form, three corner grid point IDs are required to specify a 3D element face, two corner grid point ids are required to specify a 2D element edge. For shell faces, the sequence of G1,G2,G3 of the element definition means TOP(top) and the reverse order means BTM(bottom), blanks or zeros means BOTH(both).
6. FACEIDi varies depending on element type and DIM. Refer to remark 9 for the list of FACEIDs.
7. For shell elements, TOP, BTM and BOTH are to define the contact face on top, bottom and both sides of the element. A blank is the same as BOTH. It is possible to have TOP and BTM in the same BCSURF entry but not allowed to combine BTM with BOTH or TOP with BOTH.
8. For beam elements, ENDA and ENDB express beam or bar enda and endb points respectively. A blank means the whole beam or bar body.
9. List of face ID.

Refer to following pages

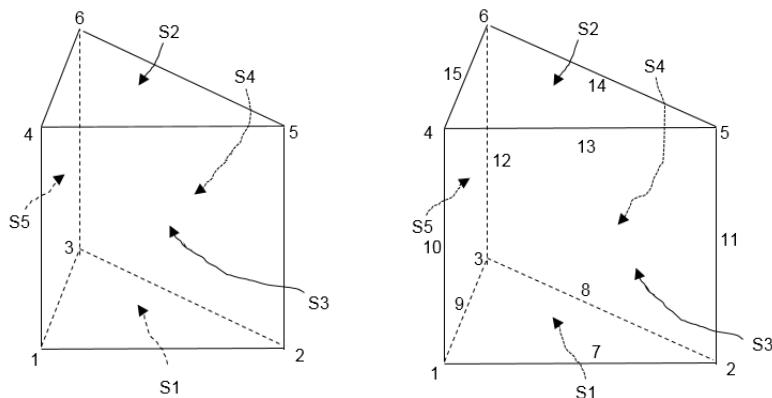
CHEXA

|    | Linear Element | Quadratic Element   |
|----|----------------|---------------------|
| S1 | 4-3-2-1        | 4-3-2-1-11-10-9-12  |
| S2 | 5-6-7-8        | 5-6-7-8-17-18-19-20 |
| S3 | 1-2-6-5        | 1-2-6-5-9-14-17-13  |
| S4 | 2-3-7-6        | 2-3-7-6-10-15-18-14 |
| S5 | 3-4-8-7        | 3-4-8-7-11-16-19-15 |
| S6 | 4-1-5-8        | 4-1-5-8-12-13-20-16 |



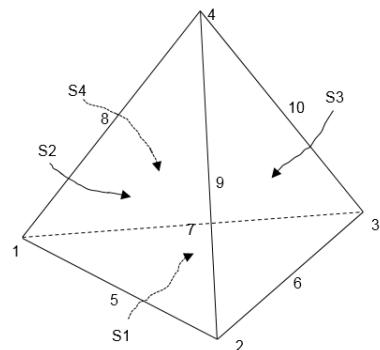
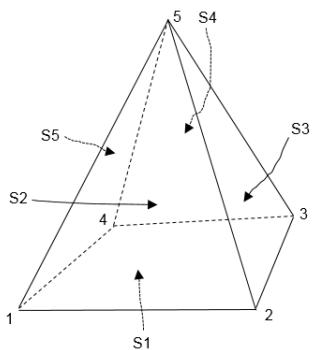
CPENTA

|    | Linear Element | Quadratic Element  |
|----|----------------|--------------------|
| S1 | 3-2-1          | 3-2-1-8-7-9        |
| S2 | 4-5-6          | 4-5-6-13-14-15     |
| S3 | 1-2-5-4        | 1-2-5-4-7-11-13-10 |
| S4 | 2-3-6-5        | 2-3-6-5-8-12-14-11 |
| S5 | 3-1-4-6        | 3-1-4-6-9-10-15-12 |



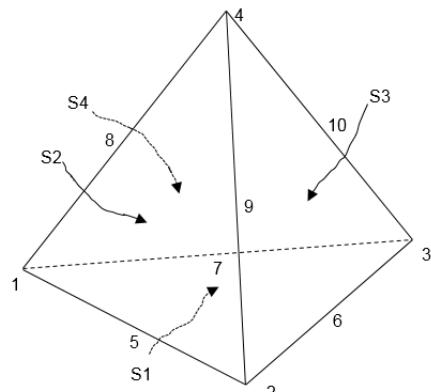
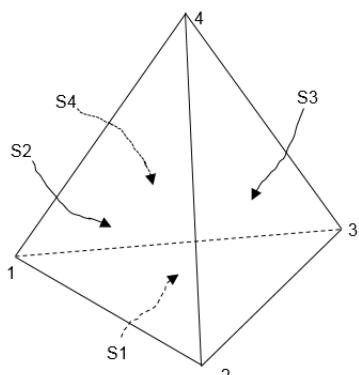
## CPYRAM

|    | Linear Element | Quadratic Element |
|----|----------------|-------------------|
| S1 | 4-3-2-1        | 4-3-2-1-8-7-6-9   |
| S2 | 1-2-5          | 1-2-5-6-11-10     |
| S3 | 2-3-5          | 2-3-5-7-12-11     |
| S4 | 3-4-5          | 3-4-5-8-13-12     |
| S5 | 4-1-5          | 4-1-5-9-10-13     |



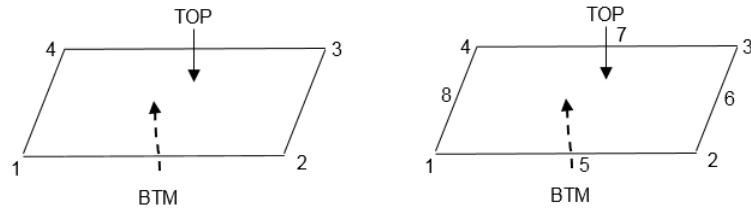
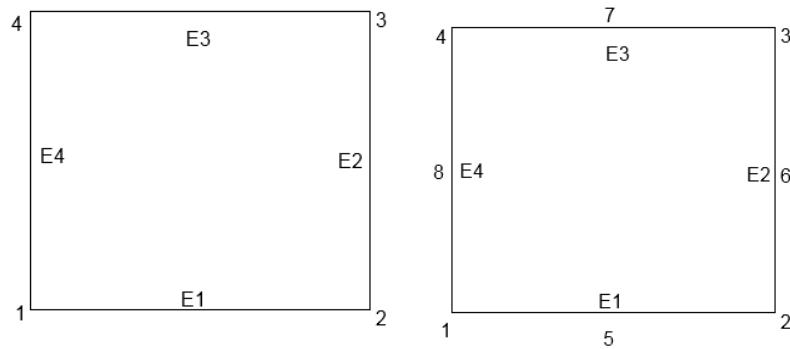
## CTETRA

|    | Linear Element | Quadratic Element |
|----|----------------|-------------------|
| S1 | 3-2-1          | 3-2-1-6-5-7       |
| S2 | 1-2-4          | 1-2-4-5-9-8       |
| S3 | 2-3-4          | 2-3-4-6-10-9      |
| S4 | 3-1-4          | 3-1-4-7-8-10      |



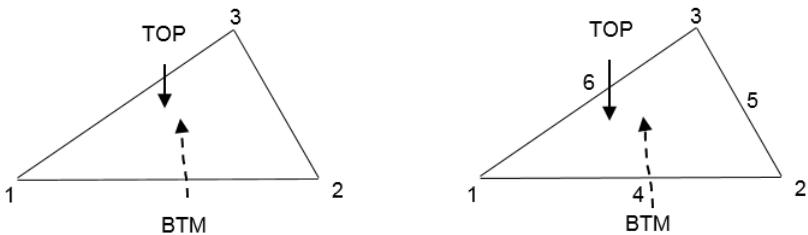
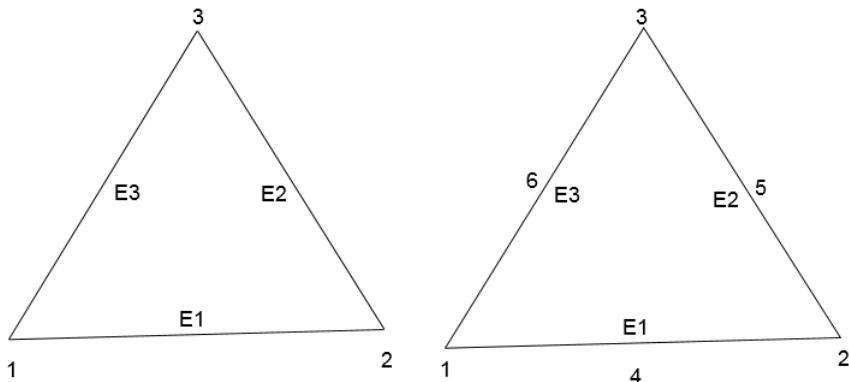
CQUAD4/CQUAD/CQUAD8/CQUADR when DIM is 3D

|     | Linear Element | Quadratic Element |
|-----|----------------|-------------------|
| E1  | 1- 2           | 1-2-5             |
| E2  | 2-3            | 2-3-6             |
| E3  | 3-4            | 3-4-7             |
| E4  | 4-1            | 4-1-8             |
| TOP | 1-2-3-4        | 1-2-3-4-5-6-7-8   |
| BTM | 4-3-2-1        | 4-3-2-1-7-6-5-8   |



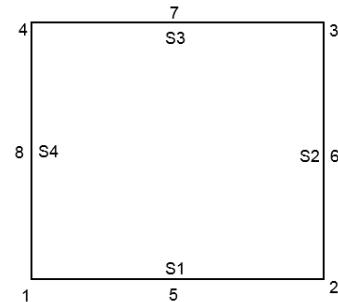
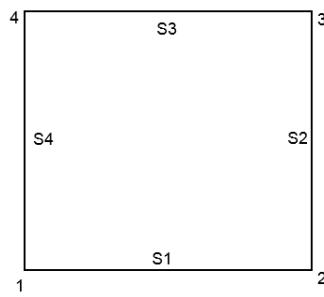
## CTRIA3/ CTRIA6/CTRIAR when DIM is 3D

|     | Linear Element | Quadratic Element |
|-----|----------------|-------------------|
| E1  | 1- 2           | 1-2-4             |
| E2  | 2-3            | 2-3-5             |
| E3  | 3-1            | 3-1-6             |
| TOP | -2-3           | 1-2-3-4-5-6       |
| BTM | 3-2-1          | 3-2-1-5-4-6       |



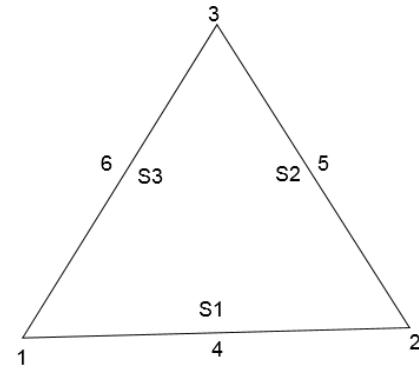
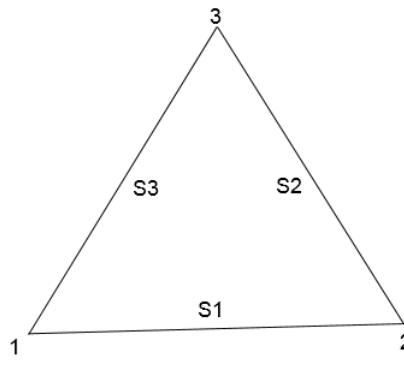
## CQUAD4, CQUAD8 and CQUADX when DIM is 2D

|    | Linear Element | Quadratic Element |
|----|----------------|-------------------|
| S1 | 1- 2           | 1-2-5             |
| S2 | 2-3            | 2-3-6             |
| S3 | 3-4            | 3-4-7             |
| S4 | 4-1            | 4-1-8             |



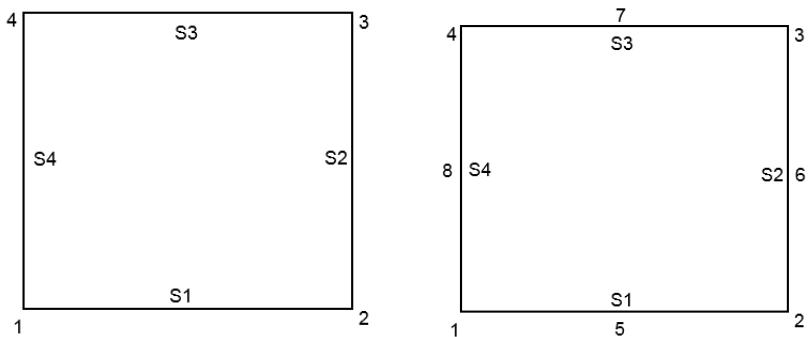
## CTRIA3, CTRIA6, CTRIAX, CTRIA6X when DIM is 2D

|    | Linear Element | Quadratic Element |
|----|----------------|-------------------|
| S1 | 1- 2           | 1-2-4             |
| S2 | 2-3            | 2-3-5             |
| S3 | 3-1            | 3-1-6             |



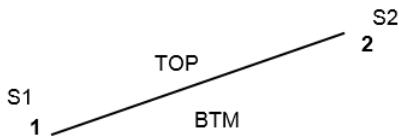
CTRIA3, CTRIA6, CTRIAX, CTRIA6 when DIM is 2D

|    | Linear Element | Quadratic Element |
|----|----------------|-------------------|
| S1 | 1- 2           | 1-2-4             |
| S2 | 2-3            | 2-3-5             |
| S3 | 3-1            | 3-1-6             |



CAXISYM

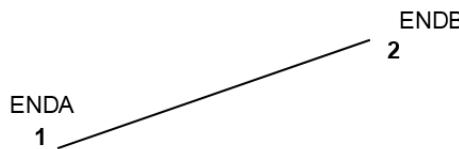
|     | Linear Element | Quadratic Element |
|-----|----------------|-------------------|
| S1  | 1              | 1                 |
| S2  | 2              | 2                 |
| TOP | 1-2            | 1-2-3             |
| BTM | 2-1            | 2-1-3             |



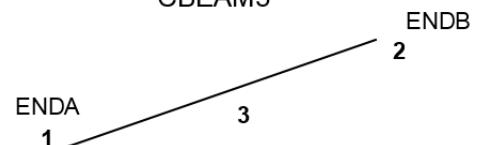
CBAR, CBEAM

|         | Linear Element | Quadratic Element |
|---------|----------------|-------------------|
| ENDA    | 1              | 1                 |
| ENDB    | 2              | 2                 |
| (BLANK) | 1-2            | 1-2-3             |

CBEAM/CBAR



CBEAM3



## BCTABLE

Defines a Contact Table for General Contact

Defines a contact table used in SOL 101, 400 and 700. SOL 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types. The BCTABL1 option is the preferred method to define a contact table.

Format: (SOLs 101 and 400 only)

| 1       | 2         | 3       | 4       | 5       | 6      | 7       | 8      | 9     | 10 |
|---------|-----------|---------|---------|---------|--------|---------|--------|-------|----|
| BCTABLE | ID        | IDSLAVE | IDMAST  | NGROUP  | COPTS  | COPTM   |        |       |    |
|         | “SLAVE”   | IDSLA1  | ERROR   | FNTOL   | FRIC   | CINTERF | IGLUE  |       |    |
|         |           | ISEARCH | ICOORD  | JGLUE   |        | DQNEAR  |        |       |    |
|         | “FBSH”    | FRLIM   | BIAS    | SLIDE   | HARDS  | COPTS1  | COPTM1 |       |    |
|         | “BKGL”    | BGSN    | BGST    | BGM     | BGN    |         |        |       |    |
|         | “SEGS”    | PENALT  | AUGDIST | TPENALT | STKSLP |         |        |       |    |
|         | “HHHB”    | HCT     | HCV     | HNC     | BNC    | EMISS   | HBL    |       |    |
|         |           | HNL     | BNL     | HGLUE   |        |         |        |       |    |
|         | “MASTERS” | IDMA1   | IDMA2   | IDMA3   | IDMA4  | IDMA5   | IDMA6  | IDMA7 |    |
|         |           | IDMA8   | IDMA9   | ...     |        |         |        |       |    |

Format: (SOL 700 only)

| 1 | 2         | 3     | 4      | 5      | 6       | 7       | 8       | 9     | 10 |
|---|-----------|-------|--------|--------|---------|---------|---------|-------|----|
|   |           |       |        |        |         |         |         |       |    |
|   |           |       |        |        |         |         |         |       |    |
|   |           |       |        |        |         |         |         |       |    |
|   |           | SIDE  | WEIGHT | MONDIS | MONDISV | INITMON | DAMPING |       |    |
|   | "MASTERS" | IDMA1 | IDMA2  | IDMA3  | IDMA4   | IDMA5   | IDMA6   | IDMA7 |    |
|   |           | IDMA8 | IDMA9  | ...    |         |         |         |       |    |

**Examples:**

|         |         |    |    |   |     |  |  |  |  |
|---------|---------|----|----|---|-----|--|--|--|--|
| BCTABLE | 2       |    |    | 3 |     |  |  |  |  |
|         | SLAVE   | 10 |    |   | 0.2 |  |  |  |  |
|         | MASTERS | 20 | 30 |   |     |  |  |  |  |
|         | SLAVE   | 20 |    |   | 0.3 |  |  |  |  |
|         | MASTERS | 10 |    |   |     |  |  |  |  |
|         | SLAVE   | 30 |    |   | 0.2 |  |  |  |  |
|         | MASTERS | 10 |    |   |     |  |  |  |  |

|         |   |   |   |   |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|
| BCTABLE | 0 | 1 | 2 | 0 |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|

| Descriptor                          | Meaning  |
|-------------------------------------|--|
| ID                                  | Identification number referenced by a BCONTACT Case Control command. See Remark 5. (Integer; Required)   |
| IDSLAVE                             | Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)   |
| IDMAST                              | Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)  |
| NGROUP                              | Flag to indicate that the continuation entries "SLAVE" and "MASTERS" are entered or not. Zero means no continuation entries are entered. Any positive integer means one or more sets of slave/master entries are entered. (Integer; Default = 1).                              |
| COPTS, COPTM,<br>COPTS1,<br>COMPTM1 | Flag to indicate how slave and master surfaces may contact. See Remark 9. (Integer; Default = 0) COPTS and COPTM apply to all slave and master surfaces on this BCTABLE. To set individual slave/master combination differently, use COPTS1 and/or COPTM1. Ignored by SOL 700. |
| "SLAVE"                             | Indicates that this line defines the touching body and its parameters.   |

| Descriptor | Meaning   |
|------------|---|
| IDSLA1     | Identification number of a BCBODY Bulk Data entry defining the touching body.<br>(Integer > 0)<br>For SOL 700, leaving IDSLA1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.   |
| ERROR      | Distance below which a node is considered touching a body. Default = blank automatic calculation. See the Bulk Data entry <a href="#">BCPARA, 1342</a> for more details.<br>Ignored by SOL 700. (Real)  |
| FNTOL      | Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. Its default value is dependent on the IBSEP value. See remark <a href="#">10</a> . Ignored by SOL 700. (Real)  |
| FRIC       | Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multidimensional table. SOL700 does not support the table definition. (Real > 0.0 or Integer > 0; Default = 0.0 for SOL400, Real > 0.0; Default=0.0 for SOL700)  |
| CINTERF    | Interference closure amount, normal to the contact surface. Default = 0. For CINTERF > 0, overlap between bodies. For CINTERF < 0., gap between bodies. Ignored by SOL 700. (Real)  |
| IGLUE      | Flag to activate glue option (Integer ≠ 0). Default is 0, no glue option. For SOL 700, IGLUE=1 is only acceptable. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, general contact methods (METHOD=TIEDxxx) can assign glue contact types as well. (See Remarks <a href="#">8</a> through <a href="#">16</a> .)                                    |
| ±1         | Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.  |
| ±2         | Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap. |
| ±3         | Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body.  |

| Descriptor | Meaning  |
|------------|--|
|            | $\pm 4$ Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.   |
| ISEARCH    | Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. Ignored by SOL 700. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0 (Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19.</li> <li>1 (Single order search) the searching order is from Slave to Master</li> <li>2 (Single order search) let the program decide which search order. See Remark 19.</li> </ul> <p>Note that ISTYP in <a href="#">BCBDPRP</a> or <a href="#">BCBODY</a> is supported with ISEARCH=0 only.</p> |
| ICOORD     | Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. Ignored by SOL 700. (Integer; Default = 0)  |
| JGLUE      | This option is only relevant if the general glue option is invoked (IGLUE > 0 and =1). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. For SOLs 101 and 400, enter 2 to activate breaking glue. (Integer; Default = 0) <p>For SOL 700 the following options are available:</p> <ul style="list-style-type: none"> <li>0 Slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.</li> </ul>  |
| DQNEAR     | Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. Ignored by SOL 700. (Real; Default = 0; which means near contact does not occur)   |
| "FBSH"     | Enter character string FBSH if the line with items FRLIM, BIAS, SLIDE, HARDS is required to change the defaults of any of these values. (SOLs 101 and 400)   |
| FRLIM      | Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$ , with $\mu$ the friction coefficient and $\sigma_n$ the contact normal stress. (Real; Default = 1.0E20)   |

| Descriptor | Meaning  |
|------------|--|
| BIAS       | Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA.<br>SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)   |
| SLIDE      | Delayed slide off distance. This entry should not be made unless ICOORD $\geq$ 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.0) |
| HARDS      | Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0)   |
| “BKGL”     | Enter the character string BKGL if the line with items BGST, BGSN, BGM, BGN is required. (SOLs 101 and 400)  |
| BGSN       | Maximum normal stress for breaking glue. See Remark 10. (Real; Default = 0.0)  |
| BGST       | Maximum tangential stress for breaking glue. See Remark 10. (Real; Default = 0.0)  |
| BGM        | Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)   |
| BGN        | Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)   |
| “SEGS”     | Enter the character string SEGS if the line with items PENALT and AUGDIST, TPENALT and STKSLP is required for the segment-to-segment contact.  |
| PENALT     | Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real $\geq$ 0.0; see Remark 16. for default)  |
| AUGDIST    | Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real $\geq$ 0, see Remark 17. for default)  |
| “HHHB”     | Enter the character string HHHB if the line with items HCT, HCV, HNC, ..., up to HGLUE is required in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.   |

| Descriptor | Meaning  |
|------------|--|
| HCT        | Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.w., when IGLUE > 0.)                |
| HCV        | Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).   |
| HNC        | Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).   |
| BNC        | Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0.) |
| EMISS      | Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables. (Real or Integer; Default = 0.0)  |
| HBL        | Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).  |
| HNL        | Heat transfer coefficient for nonlinear convective heat flow for near field behavior. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0)  |

| Descriptor               | Meaning   |
|--------------------------|---|
| BNL                      | Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0)   |
| HGLUE                    | Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. ( $0 \leq$ Integer $\leq 1$ ; Default = 0)  |
| FK<br>(SOL 700 only)     | Kinetic coefficient of friction. (Real $\geq 0.0$ ; Default = 0.0)  |
| EXP<br>(SOL 700 only)    | Exponential decay coefficient. (Real $\geq 0.0$ ; Default = 0.0)  |
| METHOD<br>(SOL 700 only) | Character, Influences the contact type used. See Remark 19. Options are:<br>FULL: Regular Contact (Default)<br>AIRBAG: Single Surface Contact<br>SS1WAY: Surface To Surface One Way<br>SS2WAY: Surface To Surface Two Way<br>RB1WAY: Rigid Body One Way To Rigid Body<br>RB2WAY: Rigid Body Two Way To Rigid Body<br>RNRB: Rigid Nodes To Rigid Body<br>TIEDNS: Tied Nodes to Surface<br>TIEDNSO: Tied Nodes to Surface with Offset<br>RELLIPS: Tied contact between grid points or rigid materials to ATB dummies.<br>See Remark 23.<br>BELT: Suited for modeling contact between a belt element and a rigid structure.<br>Master slave contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the slave nodes exactly on top of the master face. The slave node does not slide relative to the master face when the friction coefficient (FS) is set to 1E20<br>BELT1: Identical to BELT algorithm, except that the slave nodes are initially repositioned on top of the closest master face. All slave nodes initially penetrated or within a distance of INITMON from a master face, are repositioned. |

| Descriptor                | Meaning   |   |  |   |   |
|---------------------------|---|---|--|---|---|
| ADAPT<br>(SOL 700 only)   | Character, influences the contact type used.<br>Options are NO or YES.<br>Default = NO<br>When ADAPT=YES, the BCBODY entries IDMAi must be defined as:<br>behav=DEFORM<br>bsid references a BCPROP  |   |  |   |   |
| THICK<br>(SOL 700 only)   | Shell thickness scale factor. (Real $\geq 0.0$ ; Default = same as the value in DYPARAM,CONTACT,THICK)  |   |  |   |   |
| THICKOF<br>(SOL 700 only) | Artificial contact thickness offset. (Real $\geq 0.0$ ; Default = 0.0)  |   |  |   |   |
| PENV<br>(SOL 700 only)    | Overwrites the default maximum penetration distance factor. (Real $\geq 0.0$ ; Default = 1.E20)   |   |  |   |   |
| FACT<br>(SOL 700 only)    | Scale factor for the contact forces. (Real $\geq 0.0$ ; Default = 0.1)  |   |  |   |   |
| TSTART<br>(SOL 700 only)  | Time at which the contact is activated. (Real $\geq 0.0$ ; Default = 0.0)   |   |  |   |   |
| TEND<br>(SOL 700 only)    | Time at which the contact is deactivated. (Real $\geq 0.0$ ; Default = 1.e20)   |   |  |   |   |
| MAXPAR<br>(SOL 700 only)  | Maximum parametric coordinate in segment. (Real $\geq 0.0$ ; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.   |   |  |   |   |
| IADJ<br>(SOL 700 only)    | Adjacent material treatment option for solid elements. (Integer $\geq 0$ ; Default = 1) <table> <tr> <td>0</td><td>Solid element faces are included only for free boundaries.</td></tr> <tr> <td>1</td><td>Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.</td></tr> </table> | 0 | Solid element faces are included only for free boundaries. | 1 | Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact. |
| 0                         | Solid element faces are included only for free boundaries.  |   |  |   |   |
| 1                         | Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.   |   |  |   |   |
| IGNORE<br>(SOL 700 only)  | Ignore initial penetrations. (Integer $\geq 1$ ) <table> <tr> <td>1</td><td>Allow initial penetrations to exist by tracking it</td></tr> <tr> <td>2</td><td>Move nodes to eliminate initial penetrations</td></tr> </table>   | 1 | Allow initial penetrations to exist by tracking it         | 2 | Move nodes to eliminate initial penetrations  |
| 1                         | Allow initial penetrations to exist by tracking it  |   |  |   |   |
| 2                         | Move nodes to eliminate initial penetrations  |   |  |   |   |
| AUTO<br>(SOL 700 only)    | (Character)<br>Options are:<br>YES      Automatic Contacts Activated (Default)<br>NO      Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.  |   |  |   |   |

| Descriptor                | Meaning  |
|---------------------------|--|
| SIDE<br>(SOL 700 only)    | Defines which side will be the monitoring side of a master face. The opposite side of the master face will be the penetration side. See Remark 24. (Characters; default=BOTH)<br><br>BOTH: The side from which a slave node approaches the master face will become the monitoring side.<br>TOP: The monitoring side will always be on the side of the master face that its normal is pointing at.<br>BOTTOM: The monitoring side is always on the opposite side of the master face that its normal is pointing at. |
| WEIGHT<br>(SOL 700 only)  | The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH)<br><br>BOTH: $M_{scale} = \frac{M_{slave} M_{master}}{M_{slave} + M_{master}}$<br>SLAVE: $M_{scale} = M_{slave}$<br>MASTER: $M_{scale} = M_{master}$<br>NONE: $M_{scale} = 1.0$  |
| MONDIS<br>(SOL 700 only)  | Defines the fixed part of the monitoring distance. When the normal distance of a slave node to a master face becomes smaller than the monitoring distance the slave node will tag itself to the master face. The side from which the slave node is moving towards the master face becomes the monitoring region. (Character; default=FACTOR)<br><br>FACTOR: The monitoring distance is equal to a factor times a characteristic length of the master faces. The factor is specified in MONDISV.                    |
| DISTANCE                  | The monitoring distance is specified in MONDISV.   |
| :                         |  |
| MONDISV<br>(SOL 700 only) | Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)  |
| INITMON<br>(SOL 700 only) | Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real > 0.0; default=MONDIS)  |
| DAMPING<br>(SOL 700 only) | Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a slave node with respect to a master face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES)<br><br>YES damping is activated<br>N damping is not activated   |

| Descriptor | Meaning  |
|------------|--|
| "MASTERS"  | Indicates the start of the list of bodies touched by touching body IDSLA1.                   |
| IDMAi      | Identification numbers of BCBODY Bulk Data entries defining touched bodies.<br>(Integer > 0) |

**Remarks:**

1. BCTABLE defines surface contact.
2. If BCTABLE is not given, the default for contact analysis is assumed if BCONTACT=ALLBODY in Case Control Section, and, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exists if the user provides IDSLAVE and IDMAST. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSLAVE and IDMAST blank, then NGROUP is normally required and continuation entries are usually expected for NGROUP SLAVE/MASTER combinations. Exceptions are for SOL 700 self-contact may be designated using a slave IDSLA1 of zero and no MASTER entry.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY's. BCTABLE, 0 is not used to define the contact relationship for any loadcases. ONLY BCTABLE, n(n>0), will dominate the contact analysis for each loadcase. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.
6. The line starting with "HHHB" is used for heat transfer or thermal contact analyses only. Also, see Remark 17.
7. It is not necessary to enter all continuation lines between SLAVE and MASTERS. For example, if an entry is required on the 4<sup>th</sup> SLAVE line, the first 3 must be entered with some being blank and those after the 4<sup>th</sup> may be omitted. All continuation lines prior to the last needed must be entered.
8. Many slave/masters may be entered up to the number specified in Remark 5. A new slave entry may not begin until the masters from the previous entry are finished (as shown in the example). The number of master surfaces for any given slave surface is limited by the number specified in Remark 5., however most GUI's produce one slave surface and one master surface per slave/masters pairs.

9. COPTS, COPTM, COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

10. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BGSN = 0.0 the first term is ignored. If BGST is zero, the second term is ignored.

11. For hard contact, with HGLUE=1:

- a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.

- b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

Note:

“Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

12. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

13. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BBL} - T_B^{BBL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[ HCT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

14. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
15. Table IDs of tables used on the BCTABLE and the BCBODY entry must be unique.
16. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$PENALT = \frac{0.5(S_i + S_j)}{\bar{L}}$$

The body stiffness ( $S_i$  and  $S_j$ ), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length ( $\bar{L}$ ) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and given by:

$$\text{PENALT} = \frac{1000S_i}{\bar{L}}$$

17. By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

$$\text{AUGDIST} = 0.05\bar{L}$$

18. The multipoint constraint equations (MPC equations) internally created from body contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH. This capability is good for all solutions except SOL 700.
19. When ISEARCH=0 (and ISTYP=0 in default on BCBODY), the search order is from lower BCBODOY ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2

CL1= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY ID1

CL2= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY ID2

CL1 and CL2 refers to characteristic length of BCBODY ID1 and ID2.

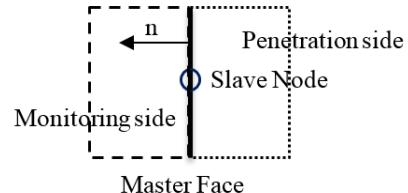
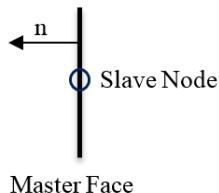
The search order is from lower BCBODY ID1 to higher BCBODY ID2 if CB1<=1.05\*CB2.

Otherwise, if CB1>1.05\*CB2 the search order is from BCBODY ID2 to BCBODY ID1.

20. Fields ISEARCH, DQNEAR and "BKGL" are not supported in segment to segment contact analysis.
21. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ( $\mu \cdot F_{NORMAL}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
22. When the initial stress-free(ICOORD=1), Node-to-Seg updates model geometry to close the gap, nodes are projected onto the contact body; but Seg-to-Seg treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
23. When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SLAVE body and BCELIPS is only available for MASTER body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable.

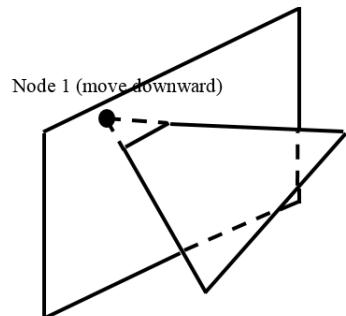
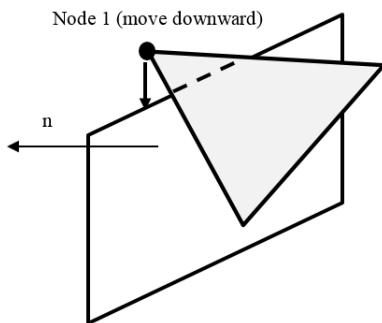
24. The options TOP/BOTTOM are useful in the following cases:

- When a slave node initially is located on the master face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the master surface is defined:  
SIDE=TOP case:



- When hooking of slave nodes on the wrong side of a master face might occur. This often is the case when the master face is at the edge of a shell element structure:

SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



25. Recommended usage of WEIGHT:

| SLAVE      | MASTER     | WEIGHT |
|------------|------------|--------|
| Deformable | Deformable | BOTH   |
| Deformable | Rigid      | SLAVE  |
| Rigid      | Deformable | MASTER |
| Rigid      | Rigid      | NONE   |

Default setting when BCELIPS is used in SLAVE or MASTER:

| SLAVE       | MASTER      | WEIGHT |
|-------------|-------------|--------|
| Non-BCELIPS | BCELIPS     | SLAVE  |
| BCELIPS     | Non-BCELIPS | NONE   |
|             | BCELIPS     |        |

**BCTABLE - Glued Option**

Defines a Contact Table for Permanently Glued Contact

Defines a glued contact used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and SOL 400 for the permanently-glued or general contact.

Format: (SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400 only)

| 1       | 2         | 3       | 4      | 5      | 6     | 7        | 8      | 9      | 10 |
|---------|-----------|---------|--------|--------|-------|----------|--------|--------|----|
| BCTABLE | ID        |         |        | NGROUP | COPTS | COPTM    |        |        |    |
|         | “SLAVE”   | IDSLA1  | ERROR  |        |       | CINTER F | IGLUE  |        |    |
|         |           | ISEARCH | ICOORD |        |       |          |        |        |    |
|         |           | “FBSH”  |        | BIAS   | SLIDE |          | COPTS1 | COPTM1 |    |
|         | “MASTERS” | IDMA1   | IDMA2  | IDMA3  | IDMA4 | IDMA5    | IDMA6  | IDMA7  |    |
|         |           | IDMA8   | IDMA9  | ...    |       |          |        |        |    |

Examples:

|         |         |      |       |      |     |    |    |    |  |
|---------|---------|------|-------|------|-----|----|----|----|--|
| BCTABLE | 101     |      |       | 2    | 61  | 61 |    |    |  |
|         | SLAVE   | 20   | 0.002 |      | 0.2 |    | 1  |    |  |
|         |         | 1    |       |      |     |    |    |    |  |
|         |         | FBSH |       | 0.01 |     |    | 60 | 60 |  |
|         | MASTERS | 31   | 32    | 33   |     |    |    |    |  |
|         | SLAVE   | 40   |       |      |     |    | 1  |    |  |
|         | MASTERS | 51   | 52    | 53   | 54  |    |    |    |  |

| Descriptor    | Meaning  |
|---------------|--|
| ID            | Identification number of a BCTABLE entry to be selected by BCONTACT Case Control command. See Remark 5. for the general contact. Case Control, BCONTACT=ALLBODY, cannot be used for the permanently glued contact. See Remark 1. (Integer $\geq 0$ ; Required) |
| NGROUP        | Flag to indicate the number of pairs of “SLAVE” and “MASTERS” entries in the continuation lines. (Integer $> 0$ ; Required)  |
| COPTS1, COPTM | Flag to indicate how slave and master surfaces may contact. See Remark 6. for the general contact.   |
| “SLAVE”       | Indicates that this line defines the slave or touching body and its parameters.  |
| IDSLA1        | Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer $> 0$ ; Required)   |

| Descriptor | Meaning   |
|------------|---|
| ERROR      | Distance below which a node is considered touching a body. When this field is left blank or 0.0, the value is taken from BCPARA = 0, ERROR, if it is specified. Otherwise, Nastran automatically calculates the value. See the Bulk Data entry <a href="#">BCPARA, 1342</a> for more details. (Real)  |
| CINTERF    | Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies, CINTERF < 0.0, gap between bodies. (Real; Default = 0.0)   |
| IGLUE      | Flag to activate the glue options. See the descriptions of various options in BCTABLE for the general contact. This field must be specified for the glued contact. See Remark 4. (Integer > 0; Required). <ol style="list-style-type: none"> <li>1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.</li> <li>2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.</li> <li>3. Insures full moment carrying glue when shells contact. This option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified.</li> <li>4. Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.</li> </ol> |
| ISEARCH    | Option for contact searching order, from a slave (touching) to a master (touched) body or visa versa, for deformable contact. See the options in BCTABLE for the general contact. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0 (Double orders search) 0(Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE.</li> <li>1 (Single order search) the searching order is from Slave to Master</li> <li>2 (Single order search) let the program decide which search order. See Remark 19. in BCTABLE.</li> </ul>   |

| Descriptor     | Meaning   |
|----------------|---|
| ICOORD         | Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)   |
| "FBSH"         | Character string used to introduce BIAS, COPTS1, and COPTM1.  |
| BIAS           | Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)   |
| SLIDE          | Delayed slide off distance. This entry should not be made unless ICOORD $\geq$ 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3- D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.0) |
| COPTS1, COPTM1 | Flag to indicate how individual pair of slave and master contact surfaces may contact. These two fields override COPTS and COPTM. See Remark 6.   |
| "MASTERS"      | Indicates the start of the list of bodies touched by touching body IDSLA1.  |
| IDMAi          | Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0; Required)   |

**Remarks:**

1. This BCTABLE entry is tailored specifically for what we called the permanently-glued contact (or tied contact), from the entry of BCTABLE for the general contact, where the glued is an option in the approaching-and-touched contact. Most of the descriptions and remarks on the fields in this entry can be found in the BCTABLE for the general contact.
2. Permanent glued contact for small rotation condition is defined when all IGLUE fields of BCTABLE, reference by the very first loadcase, (the 1st STEP of the 1st SUBCASE in SOL 400 or the 1st SUBCASE in others) is set to larger than 0, IGLUE(>0). In BCPARA with ID=0, the value of NLGLUE may not be 1.

For large deformation and large rotation condition, permanent glued contact is activated by negative IGLUE (<0) and works in corresponding subcase or step in the corresponding subcase or step where it is defined, no matter what is set (1 or 0). This type of permanent glued contact with large deformation and large rotation may be applied together with combination of general glued contact and touching contact. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.

3. The MPC equations internally created for the glued contact can be printed out in Nastran standard punch file by using Case Control command, NLOPRM MPCPCH=BEGN.

4. IGLUE=±2 or ±4 is favorable to passing the GROUNDCHECK. The initial stress-free contact is also available to preserve the six rigid-body modes with IGLUE = 1 or 3.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY's. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.
6. COPTS, COPTM COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA)

7. It is not supported that a permanently-glued contact is used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the general contact with glued option must be performed.

**BCTABLE**

Defines a Contact Table in SOL 600

Defines a contact table in SOL 600.

For heat transfer or thermal contact do not enter parameter PARAM,MRCONVER,11 in the bulk data. For structural analysis, enter PARAM,MRCONVER,11 if SLAVE lines "FBSH" or beyond are used.

**Format:**

| 1       | 2         | 3       | 4      | 5       | 6     | 7       | 8      | 9      | 10 |
|---------|-----------|---------|--------|---------|-------|---------|--------|--------|----|
| BCTABLE | ID        | IDSLAVE | IDMAST | NGROUP  |       |         |        |        |    |
|         | "SLAVE"   | IDSLA1  | ERROR  | FNTOL   | FRIC  | CINTERF | IGLUE  |        |    |
|         |           | ISEARCH | ICOORD | JGLUE   | TOLID | DQNEAR  | DISTID |        |    |
|         |           | "FBSH"  | FRLIM  | BIAS    | SLIDE | HARDS   | COPTS1 | COPTM1 |    |
|         |           | "BKGL"  | BGSN   | BGST    | BGM   | BGN     |        |        |    |
|         |           | "HHHB"  | HCT    | HCV     | HNC   | BNC     | EMISS  | HBL    |    |
|         |           | "SEGS"  | PENALT | AUGDIST |       |         |        |        |    |
|         | "MASTERS" | IDMA1   | IDMA2  | IDMA3   | IDMA4 | IDMA5   | IDMA6  | IDMA7  |    |
|         |           | IDMA8   | IDMA9  | ...     |       |         |        |        |    |

**Examples:**

|         |         |    |    |   |     |  |  |  |  |
|---------|---------|----|----|---|-----|--|--|--|--|
| BCTABLE | 2       |    |    | 3 |     |  |  |  |  |
|         | SLAVE   | 10 |    |   | 0.2 |  |  |  |  |
|         | MASTERS | 20 | 30 |   |     |  |  |  |  |
|         | SLAVE   | 20 |    |   | 0.3 |  |  |  |  |
|         | MASTERS | 10 |    |   |     |  |  |  |  |
|         | SLAVE   | 30 |    |   | 0.2 |  |  |  |  |
|         | MASTERS | 10 |    |   |     |  |  |  |  |

|         |   |   |   |   |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|
| BCTABLE | 0 | 1 | 2 | 0 |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|

| Descriptor             | Meaning  |
|------------------------|--|
| ID                     | Identification number referenced by a BCONTACT Case Control command. See Remark 6. (Integer; Required) |
| IDSLAVE<br>(3,1) [3,1] | Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)             |
| IDMAST<br>(5,1) [15,1] | Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)    |

| Descriptor             | Meaning  |
|------------------------|--|
| NGROUP<br>(2,1) [2,1]  | Flag to indicate that the continuation entries “SLAVE” and “MASTERS” are entered or not. Zero means no continuation entries are entered. Any positive integer means one or more sets of slave/master entries are entered. (Integer; Default = 1).  |
| COPTS1,<br>COMPTM1     | Flag to indicate how slave and master surfaces may contact. See Remark 10. (Integer; Default = 0).   |
| “SLAVE”                | Indicates that this line defines the touching body and its parameters.   |
| IDSLA1<br>(3,1) [3,1]  | Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0)  |
| ERROR<br>(3,2) [3,2]   | Distance below which a node is considered touching a body. (Real for BCTABLE with ID = 0, Real or Integer for BCTABLE with ID > 0). If the value is an Integer, it is the ID of a TABLED1 or TABL3Di providing error vs time or other variables. Default = blank automatic calculation of ERROR without any table dependence. See the Bulk Data entry <a href="#">BCPARA, 1349</a> for more details. |
| FNTOL<br>(3,3) [5,1]   | Separation force or stress (depending BCPARA IBSEP) on above which a node separates from a body. Default is maximum residual force. (Real)   |
| FRIC<br>(3,4) [5,2]    | Friction coefficient. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real > 0.0 or Integer; Default = 0.0)   |
| CINTERF<br>(3,5) [5,3] | Interference closure amount, normal to the contact surface. CINTERF > 0.0, designates an overlap between bodies, CINTERF < 0.0 designates a gap between bodies. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing interference closure vs time or friction vs other variables. (Real or Integer; Default = 0.0)   |

| Descriptor             | Meaning  |
|------------------------|--|
| IGLUE<br>(3,7) [3,7]   | <p>Flag to activate glue option (Integer <math>\geq 0</math>). Default is 0, no glue option,</p> <ul style="list-style-type: none"> <li>1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.</li> <li>2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.</li> <li>3. Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body.</li> <li>4. Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.</li> </ul> |
| ISEARCH<br>(3,8) [3,8] | <p>Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. ISEARCH is not supported in segment-to-segment contact. (Integer; Default = 0)</p> <ul style="list-style-type: none"> <li>0 Double orders search) the search order is from lower BCBODDY ID's to higher ones first. Then it searches the opposite order. See Remark <a href="#">19</a>. in BCTABLE.</li> <li>1 (Single order search) the searching order is from Slave to Master</li> <li>2 (Single order search) let the program decide which search order. See Remark <a href="#">19</a>. in BCTABLE.</li> </ul>   |
| ICOORD<br>(3,9) [3,9]  | Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)  |
| JGLUE<br>(3,10) [3,10] | This option is only relevant if the glue option is invoked (IGLUE > 0). Enter 0 if a node should not separate (Default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. (Integer; Default = 0)  |
| TOLID<br>(4,2) [4,2]   | Enter 2 to activate breaking glue.   |
|                        | Contact tolerance table ID of a TABLED <i>i</i> . Used in heat transfer analysis only. (Integer; Default = 0 which means no table ID)  |

| Descriptor            | Meaning   |
|-----------------------|---|
| DQNEAR<br>(3,3) [3,3] | Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. Used in heat transfer analysis only. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real; or integer; Default = 0.0; which means near contact does not occur)   |
| DISTID<br>(4,3) [4,3] | Contact near distance table ID of a TABLED1. Used in heat transfer only. (Integer; Default = 0 which means near contact does not occur) (Integer; Default = 0 which means no table ID)  |
| "FBSH"                | Enter character string FBSH if the line with items FRLIM, BAIS, SLIDE, HARDS is required to change the defaults of any of these values.   |
| FRLIM<br>[5,4]        | Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$ , with $\mu$ the friction coefficient and $\sigma_n$ the contact normal stress. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real or integer; Default = 1.0E20) |
| BIAS<br>[5,5]         | Contact tolerance bias. A non-blank entry will override the BIAS entered on the BCBODY entry. (Real value between 1.0E-10 and 1.0; Default = 0.9)   |
| SLIDE<br>[5,6]        | Delayed slide off distance. This entry should not be made unless ICOORD $\geq$ 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default)  |
| HARDS<br>[5,7]        | Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0)  |
| "BKGL"                | Enter the character string BKGL if the line with items BGST, BGSN, BGM, BGN is required.  |
| BGSN                  | Maximum normal stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 0.0)  |
| BGST                  | Maximum tangential stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 0.0)  |

| Descriptor           | Meaning  |
|----------------------|--|
| BGM                  | First exponent for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 2.0)  |
| BGN                  | Second exponent stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 2.0)  |
| "HHHB"               | Enter the character string HHHB if the line with items HCT, HCV, HNC, etc. is required.  |
| HCT<br>[7,1]/[8,1]   | Contact heat transfer coefficient. If real, the value entered is the contact heat transfer coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the contact heat transfer coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem.)   |
| HCV<br>[7,2]/[8,2]   | Convection coefficient for near field behavior. If real, the value entered is the near field convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the near field convection coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).  |
| HNC<br>[7,3]/[8,3]   | Natural convection coefficient for near field behavior. If real, the value entered is the near field natural convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the near field natural convection coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).  |
| BNC<br>[7,4]/[8,4]   | Exponent associated with the natural convection coefficient for near field behavior. If real, the value entered is the exponent associated with near field natural convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the exponent associated with the near field natural convection coefficient vs temperature. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem.) |
| EMISS<br>[7,5]/[8,5] | Emissivity for radiation to the environment or near thermal radiation. If real, the value entered is the emissivity. If integer, the value entered is the ID of a TABLEMi entry specifying the emissivity vs temperature. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem.)   |
| HBL<br>[7,6]/[8,6]   | Separation distance dependent thermal convection coefficient. If real, the value entered is the separation distance dependent thermal convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the separation distance dependent thermal convection coefficient. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).   |
| "SEGS"               | Enter the character string SEGS if segment-to-segment contact is desired (note that for MSC Nastran 2012 segment-to-segment contact cannot include friction.)  |

| Descriptor            | Meaning  |
|-----------------------|--|
| PENALT                | Penalty factor (stiffness) used for segment-to-segment contact analysis. (Real; Default = leave blank, meaning the program will calculate the value)     |
| AUGDIST               | Penetration distance above which additional penalty stiffness will be added. (Real; Default = leave blank, meaning the program will calculate the value) |
| “MASTERS”             | Indicates the start of the list of bodies touched by touching body IDSLA1.   |
| IDMAi<br>(4,i) [15,1] | Identification numbers of BCBODY Bulk Data entries defining touched bodies.<br>(Integer > 0)   |

### Remarks:

1. BCTABLE defines surface contact.
2. If BCTABLE is not given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exists if the user provides IDSLAVE and IDMAST. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSLAVE and IDMast blank, then NGROUP is normally required and continuation entries are usually expected for NGROUP SLAVE/MASTER combinations. An exception if no contact is desired in increment zero or a particular subcase, fields 1 and 2 of the primary BCTABLE entry for that subcase are entered, all other fields left blank and no continuation lines are entered. The no contact condition may be achieved in either of two ways - set Case Control BCONTACT=ID and enter a matching BCTABLE with that ID in field 2 and all other fields blank or set BCONTACT=NONE and do not enter BCTABLE for that subcase.
5. The maximum number of slave-master combinations is 99 for versions prior to MSC Nastran 2005 and 999 for current versions. For MSC Nastran 2005 r3 and subsequent versions, the limit is 999.
6. The BCTABLE with ID=0 will be used in “increment 0”. This phase of Marc is purely elastic and is sometimes used to move rigid contact bodies so that they just touch flexible contact bodies. If no BCTABLE entries are desired in “increment 0”, do not enter any BCTABLEs with ID=0. In the examples shown, the first BCTABLE is used in subcase 2 and the second BCTABLE is used in phase 0.
7. The BCTABLE formats (primary or alternate) shown in the *MSC Nastran 2005 r3 Quick Reference Guide* may still be used in which case PARAM,MRCONVER,11 should not be entered if the “primary” format shown is used.

8. Many slave/masters may be entered up to the number specified in Remark 5. A new slave entry may not begin until the masters from the previous entry are finished (as shown in the example). The number of master surfaces for any given slave surface is limited by the number specified in Remark 5., however most GUI's produce one slave surface and one master surface per slave/masters pairs.
9. (i,j) indicates the ith datablock jth field of MARC's CONTACT TABLE (without tables) history definition. [i,j] indicates Marc's Contact Table (with Tables).
10. COPTS, COPTM COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)

- = 2: only bottom faces will be in the contact description, thickness offset will be included

- = 3: only bottom faces will be in the contact description, shell thickness will be ignored

- = 4: only top faces will be in the contact description, thickness offset will be included

- = 5: only top faces will be in the contact description, shell thickness will be ignored

- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

- Note: For flexible bodies, B=6 for both bodies in a contact combination can only be used for glued contact. If the contact bodies are not glued and both bodies specify B=6, a FATAL ERROR will be issued. Should this fatal error occur, please choose a different B option and re-run the analysis.

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)

- = 10: only the free and hard shell edges are included in the contact description

- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA)

11. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BSGN = 0.0 the first term is ignored. If BGST is zero, the second term is ignored.

12. If the "SEGS" entry is used, "CTYPE" on BCBODY must also be entered and parameters PARAM,MRCONVER,11 and PARAM,MRCONTAB,11 must be entered.

**BCTABL1**

## Defines a Contact Table

BCTABL1 specifies a list of contact pairs through the BCNECT option for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types](#) of *Nastran SOL 400 Getting Started Guide*). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103, 105, 107 - 112 and standard 200 can only support permanent glue. When SOL 400 optimization, it can support all contact types. This contact table is activated in the BCONTACT Case Control command.

Format:

| 1       | 2    | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|------|-----|--------|-----|-----|-----|-----|-----|----|
| BCTABL1 | BCID | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|         | ID8  | ID9 | -etc.- |     |     |     |     |     |    |

Examples:

|         |    |     |    |    |   |     |    |    |  |
|---------|----|-----|----|----|---|-----|----|----|--|
| BCTABL1 | 2  | 198 | 62 | 75 | 8 | 159 | 31 | 82 |  |
|         | 17 |     |    |    |   |     |    |    |  |

|         |   |    |    |  |  |  |  |  |  |
|---------|---|----|----|--|--|--|--|--|--|
| BCTABL1 | 0 | 23 | 56 |  |  |  |  |  |  |
|---------|---|----|----|--|--|--|--|--|--|

|         |      |      |    |      |    |     |      |     |  |
|---------|------|------|----|------|----|-----|------|-----|--|
| BCTABL1 | 5    | 31   | 35 | THRU | 40 | 163 | THRU | 169 |  |
|         | 1077 | 1088 |    |      |    |     |      |     |  |

| Describer | Meaning   |
|-----------|---|
| BCID      | Unique identification number referenced by a BCONTACT Case Control command.<br><br>See Remark 3. (Integer $\geq 0$ ; default=0) |
| IDi       | Identification number of BCNECT entry (Integer $> 0$ or “THRU”; ID1 is required).<br>See Remark 4. and 5.                       |

Remarks:

1. BCTABL1 defines surface contact. If BCTABL1 does not exist, the Case Control command BCONTACT=BCID may refer to the BCNECT Bulk Data entry directly.
2. If neither BCTABL1 nor BCNECT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABL1 or BCNECT is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABL1 or BCNECT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.

3. For SOLs 101 and 400, the BCTABL1 or BCONECT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY1's. To place an entry in any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or BCONECT=BCID. When BCONTACT=ALLBODY, there is no BCID of BCTABL1 or BCONECT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.
4. When the "THRU" option is used, all BCONECT entries associated with intermediate BCONECT IDs must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**BCTRIM****Defines the Geometry of a Trimming Curve**

Defines the geometry of a trimming curve used to specify a NURBS for a rigid contact body.

**Format:**

| 1      | 2       | 3          | 4          | 5          | 6          | 7      | 8 | 9 | 10 |
|--------|---------|------------|------------|------------|------------|--------|---|---|----|
| BCTRIM | IDtrim  | NPTUtrim   | NORUtrim   | NSUBtrim   |            |        |   |   |    |
|        | "COORD" | Xisoparam1 | Yisoparam1 | Xisoparam2 | Yisoparam2 | -etc.- |   |   |    |
|        | "HOMO"  | Homot1     | Homot2     | Homot3     | -etc.-     |        |   |   |    |
|        | "KNOT"  | Knott1     | Knott2     | Knott3     | -etc.-     |        |   |   |    |

**Examples:**

|        |       |     |     |     |     |  |  |  |  |
|--------|-------|-----|-----|-----|-----|--|--|--|--|
| BCTRIM | 202   | 2   | 2   | 50  |     |  |  |  |  |
|        | COORD | 0.0 | 0.0 | 1.0 | 0.0 |  |  |  |  |
|        | HOMO  | 1.  | 1.  |     |     |  |  |  |  |
|        | KNOT  | 0.0 | 0.0 | 1.  | 1.  |  |  |  |  |

| Descriptor  | Meaning  |
|---|--|
| IDtrim  | Unique trimming vector identification number referenced by a BCNURBS entry. (Integer > 0)  |
| NPTUtrim  | Number of control points for this trimming vector. (Integer > 0)   |
| NORUtrim  | Order for this trimming vector. (Integer > 0)  |
| NSUBtrim  | Number of subdivisions for this trimming vector. (Integer > 0)   |
| "COORD"   | Indicate the start of the list of isoparametric coordinates of points.   |
| Xisoparam1,<br>Yisoparam1,<br>Xisoparam2,<br>Yisoparam2, etc. | First and second coordinates of point in isoparametric space. There must be NPTUtrim set of (Xisoparam, Yisoparam) entries. (Real) |
| "HOMO"  | Indicate the start of the list of homogeneous coordinates of this trimming vector.   |
| Homot1, Homot2,<br>Homot3, etc.                               | Homogeneous coordinates of this trimming vector. There must be NPTUtrim entrie. (Real; 0.0 to 1.0)                                 |
| "KNOT"  | Indicate the start of the list of knot vectors of this trimming vector.  |
| Knot1, Knot2,<br>Knot3, etc.                                  | Knot vectors of this trimming vector. There must be (NPTUtrim+NORUtrim) entries. (Real; 0.0 to 1.0)                                |

**Remarks:**

1. BCTRIM is referenced by a BCNURBS entry to define the trimming curve of a NURBS for a rigid contact body.

**BDYLIST**

## Fluid Boundary List

Defines the boundary between a fluid and a structure.

**Format:**

| 1       | 2    | 3      | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|---------|------|--------|------|------|------|------|------|------|----|
| BDYLIST | RHO  | IDF1   | IDF2 | IDF3 | IDF4 | IDF5 | IDF6 | IDF7 |    |
|         | IDF8 | -etc.- |      |      |      |      |      |      |    |

**Example:**

|         |      |      |     |     |     |     |     |     |  |
|---------|------|------|-----|-----|-----|-----|-----|-----|--|
| BDYLIST | .037 | AXIS | 432 | 325 | 416 | 203 | 256 | 175 |  |
|---------|------|------|-----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning   |
|------------|---|
| RHO        | Fluid mass density at boundary. (Real $\geq 0.0$ or blank; Default is DRHO on the AXIF entry)           |
| IDF1       | Identification number of a RINGFL entry. (Integer $> 0$ or Character = “AXIS” or “LAXIS”) See Remark 7. |
| IDF2-IDFi  | Identification number of additional RINGFL entries. (Unique Integers $> 0$ )                            |

**Remarks:**

1. This entry is allowed only if an AXIF entry is also present.
2. Each entry defines a boundary if  $RHO \neq 0.0$ . The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. There is no limit to the number of BDYLIST entries specified. If the fluid density varies along the boundary, there must be one BDYLIST entry for each interval between fluid points.
5. The BDYLIST entry is not required and should not be used to specify a rigid boundary where structural points are not defined. Such a boundary is automatically implied by the omission of a BDYLIST.
6. If  $RHO=0.0$ , no boundary matrix terms will be generated to connect the GRIDB points to the fluid. This option is a convenience for structural plotting purposes. GRIDB points may be located on a fluid ring (RINGFL entry) only if the rings are included in a BDYLIST.
7. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS. See Remark 2.

**BDYOR****CHBDYi Entry Default Values**

Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.

**Format:**

| 1     | 2    | 3      | 4      | 5       | 6       | 7 | 8   | 9  | 10 |
|-------|------|--------|--------|---------|---------|---|-----|----|----|
| BDYOR | TYPE | IVIEWF | IVIEWB | RADMIDF | RADMIDB |   | PID | GO |    |
|       | CE   | E1     | E2     | E3      |         |   |     |    |    |

**Example:**

|       |       |   |   |   |   |  |    |  |  |
|-------|-------|---|---|---|---|--|----|--|--|
| BDYOR | AREA4 | 2 | 2 | 3 | 3 |  | 10 |  |  |
|-------|-------|---|---|---|---|--|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| TYPE       | Default surface type. See Remark 2. (Character)   |
| IVIEWF     | Default identification number of front VIEW entry. (Integer > 0 or blank)   |
| IVIEWB     | Default identification number of back VIEW entry. (Integer > 0 or blank)  |
| RADMIDF    | Default identification number of a RADM entry for front face. (Integer $\geq 0$ or blank)   |
| RADMIDB    | Default identification number of a RADM entry for back face. (Integer $\geq 0$ or blank)  |
| PID        | Default PHBDY property entry identification number. (Integer > 0 or blank)  |
| GO         | Default orientation grid point. (Integer $\geq 0$ ; Default = 0)  |
| CE         | Default coordinate system for defining the orientation vector. (Integer $\geq 0$ or blank)  |
| E1, E2, E3 | Default components of the orientation vector in coordinate system CE. The origin of this vector is grid point G1 on a CHBDYP entry. (Real or blank) |

**Remarks:**

- Only one BDYOR entry may be specified in the Bulk Data Section.
- TYPE specifies the type of CHBDYi element surface; allowable values are: POINT, LINE, REV, AREA3, AREA4, ELCYL, FTUBE, AREA6, AREA8, and TUBE.
- IVIEWF and IVIEWB are specified for view factor calculations only (see VIEW entry).
- GO is only used from BDYOR if neither GO nor the orientation vector is defined on the CHBDYP entry and GO is  $> 0$ .
- E1, E2, E3 is not used if GO is defined on either the BDYOR entry or the CHBDYP entry.

**BEADVAR****Topography Design Variable**

Defines design region for topography (bead or stamp) optimization.

**Format:**

| 1       | 2        | 3       | 4     | 5  | 6   | 7   | 8   | 9     | 10 |
|---------|----------|---------|-------|----|-----|-----|-----|-------|----|
| BEADVAR | ID       | PTYPE   | PID   | MW | MH  | ANG | BF  | SKIP  |    |
|         | "DESVAR" | NORM/XD | YD    | ZD | CID | XLB | XUB | DELXV |    |
|         | "GRID"   | NGSET   | DGSET |    |     |     |     |       |    |

**Example using NORM:**

|         |    |        |     |      |      |      |  |  |  |
|---------|----|--------|-----|------|------|------|--|--|--|
| BEADVAR | 10 | PSHELL | 101 | 10.0 | 20.0 | 70.0 |  |  |  |
|---------|----|--------|-----|------|------|------|--|--|--|

**Example using "DESVAR" and "GRID":**

|         |        |        |     |      |      |      |     |      |  |
|---------|--------|--------|-----|------|------|------|-----|------|--|
| BEADVAR | 10     | PSHELL | 101 | 10.0 | 20.0 | 70.0 |     | NONE |  |
|         | DESVAR | 2.0    | 3.0 | 4.0  |      | -1.0 | 1.0 |      |  |
|         | GRID   | 100    |     |      |      |      |     |      |  |

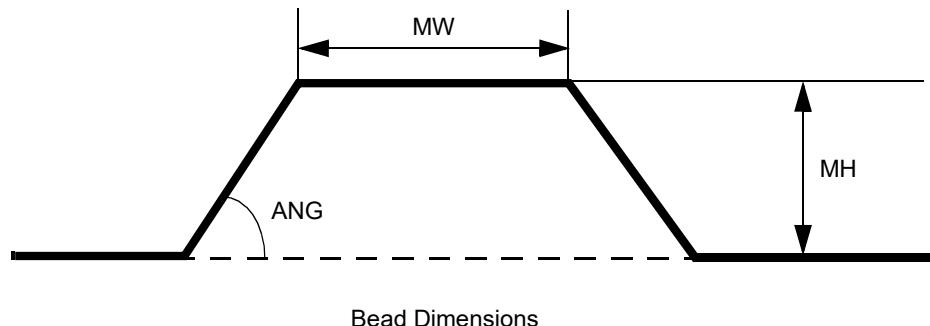
| Descriptor | Meaning   |
|------------|---|
| ID         | Unique topography design region identification number. (Integer > 0)  |
| PTYPE      | Property entry type. Used with PID to identify the element nodes to be designed. (Character: "PSHELL", "PSHEAR", "PCOMP", or "PCOMPG".)   |
| PID        | Property entry identifier. See Remark 1. (Integer > 0)  |
| MW         | Minimum bead width. This parameter controls the width of the beads. The recommended value is between 1.5 and 2.5 times the average element width. See Remark 2. (Real > 0.0)                          |
| MH         | Maximum bead height (Real > 0.0). This parameter sets the maximum height of the beads when XUB=1.0 (as Default). See Remark 2.  |
| ANG        | Draw angle in degrees (0.0 < Real < 90.0). This parameter controls the angle of the sides of the beads. The recommended value is between 60 and 75 degrees.   |
| BF         | Buffer zone ('yes' or 'no'; Default='yes'). This parameter creates a buffer zone between elements in the topography design region and elements outside the design region when BF='yes'. See Remark 3. |

| Descriptor      | Meaning  |
|-----------------|--|
| SKIP            | Boundary skip (“bc”, “load”, “both”, or “none”; Default = “both”). This parameter indicates which element nodes are excluded from the design region. “bc” indicates all nodes referenced by “SPC” and “SPC1” are omitted from the design region. “load” indicates all nodes referenced by “FORCE”, “FORCE1”, “FORCE2”, “MOMENT”, “MOMENT1”, “MOMENT2”, and “SPCD” are omitted from the design region. “both” indicates nodes with either “bc” or “load” are omitted from the design region. “none” indicates all nodes associated with elements referencing PID specified in field 4 are in the design region. |
| “DESVAR”        | Indicates that this line defines bead design variables that are automatically generated.   |
| NORM/XD, YD, ZD | Bead vector (draw direction). Norm indicates the shape variables are created in the normal directions to the elements. If XD, YD, and ZD are provided, the shape variables are created in the direction specified by the xyz vector defied by XD/YD/ZD that is given in the basic coordinate system or CID. See Remark 4. (Character or Real; Default = blank = norm).   |
| CID             | Coordinate system ID used for specifying draw direction (Blank or Integer > 0; Default = blank = basic coordinate system)  |
| XLB             | Lower bound. (Real < XUB or blank; Default = blank=0.0). This ensures the lower bound on grid movement equal to XLB*MH. See Remark 5.  |
| XUB             | Upper bound. (Real > XLB or blank; Default = 1.0). This sets the upper bound of the beads equal to XUB*MH. See Remark 5.   |
| DELXV           | Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)   |
| “GRID”          | Indicates this line defines what element nodes can be added and/or removed from topography design regions.   |
| NGSET           | All grids listed on a Bulk Data entry SET1 = NGSET are removed from topography design regions.   |
| DGSET           | All grids listed on a Bulk Data entry SET1 = DGSET are added to topography design regions.   |

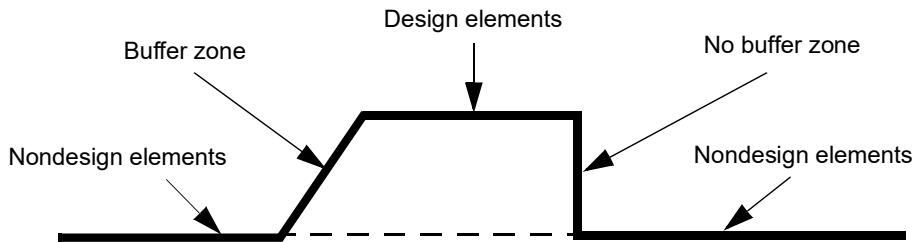
**Remarks:**

1. Multiple BEADVAR's are allowed in a single file. Combined topometry, topology, topography, sizing, and shape optimization is supported in a single file.

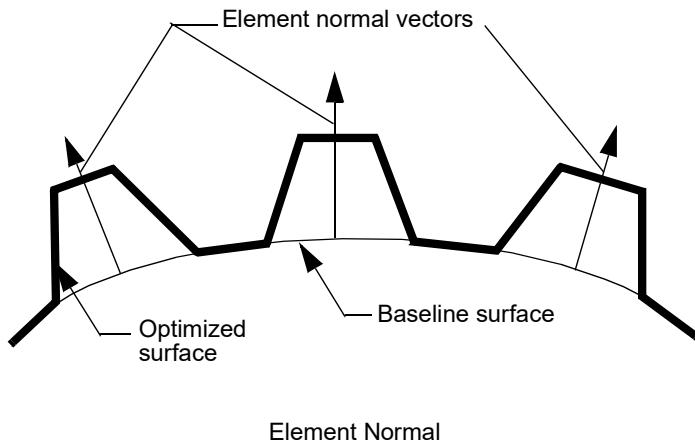
2. The user can provide allowable bead dimensions.

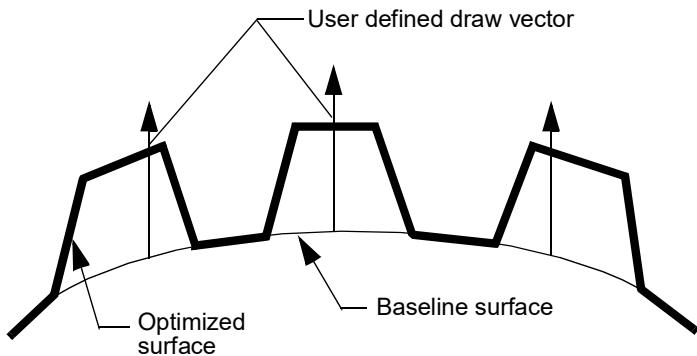


3. It is recommended to set buffer zone = yes to maintain a good quality of mesh during topography optimization.

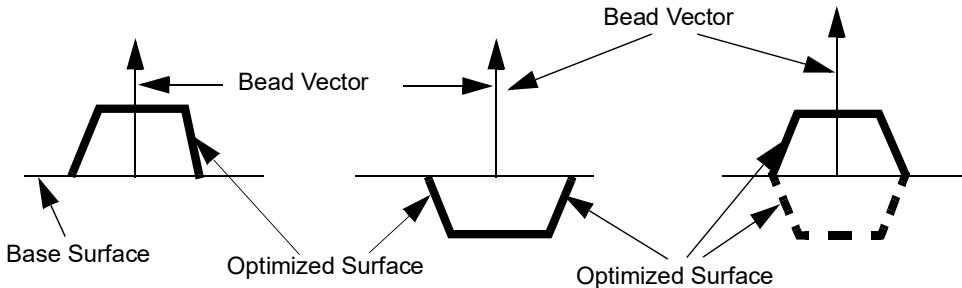


4. The grids moves in the normal direction. All element grids referenced by one BEADVAR entry must follow the right hand rule.



**User's Provided Draw Direction**

5. To force the grids to move only in the positive bead vector direction (one side of the surface), use  $XLB = 0.0$ . To force the grids to move only in the negative bead vector direction (another side of the surface), use  $XUB = 0.0$ . To allow grids to move in both positive and negative bead vector directions, use  $XLB < 0.0$  and  $XUB > 0.0$ . For example,



(a)  $XLB = 0.0$  and  $XUB = 1.0$       (b)  $XLB = -1.0$  and  $XUB = 0.0$       (c)  $XLB = -1.0$  and  $XUB = 1.0$

6. The jobname.h5 file (created by specifying MDLPRM,HDF5,0 in the bulk data input) has topology optimization results (shape changes) that can be viewed in Patran. The text file jobname.pch also has updated grid coordinates that can be copied to the original file, replace the original grids, and imported to Patran and other post-processors to view topography optimization results.
7. The BEADVAR entry cannot be used with thermal loads.

**BEAMOR****CBEAM Entry Default Values**

Defines default values for field 3 and fields 6 through 9 of the CBEAM entry.

**Format:**

| 1      | 2 | 3   | 4 | 5 | 6  | 7  | 8  | 9    | 10 |
|--------|---|-----|---|---|----|----|----|------|----|
| BEAMOR |   | PID |   |   | X1 | X2 | X3 | OFFT |    |

**Example:**

|        |  |    |  |  |     |     |       |     |  |
|--------|--|----|--|--|-----|-----|-------|-----|--|
| BEAMOR |  | 39 |  |  | 0.6 | 2.9 | -5.87 | GOG |  |
|--------|--|----|--|--|-----|-----|-------|-----|--|

**Alternate Format and Example:**

|        |  |     |  |  |    |  |  |      |  |
|--------|--|-----|--|--|----|--|--|------|--|
| BEAMOR |  | PID |  |  | G0 |  |  | OFFT |  |
|--------|--|-----|--|--|----|--|--|------|--|

|        |  |    |  |  |    |  |  |     |  |
|--------|--|----|--|--|----|--|--|-----|--|
| BEAMOR |  | 39 |  |  | 86 |  |  | GOG |  |
|--------|--|----|--|--|----|--|--|-----|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number of the PBEAM or PBCOMP entry. (Integer > 0 or blank)  |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)   |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ , using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBEAM entry) |
| OFFT       | Offset vector interpretation flag. See Remark 5. (Character or blank)  |

**Remarks:**

1. The contents of fields on this entry will be assumed for any CBEAM entry with corresponding fields that are blank.
2. Only one BEAMOR entry is allowed.
3. For an explanation of beam element geometry, see the CBEAM entry description.
4. If X1 or G0 is integer, G0 is used. If X1 or G0 is blank or real, then X1, X2, X3 is used.
5. For p-version CBEAM elements, field 9 contains the value of the built-in twist measured in radians. The OFFT option cannot be used for p-elements. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate

systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

| String | Orientation Vector | End A Offset | End B Offset |
|--------|--------------------|--------------|--------------|
| GGG    | Global             | Global       | Global       |
| BGG    | Basic              | Global       | Global       |
| GGO    | Global             | Global       | Offset       |
| BGO    | Basic              | Global       | Offset       |
| GOG    | Global             | Offset       | Global       |
| BOG    | Basic              | Offset       | Global       |
| GOO    | Global             | Offset       | Offset       |
| BOO    | Basic              | Offset       | Offset       |

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character “O” in the table replaces the obsolete character “E”.)

**BFRIC****Contact Friction**

Defines frictional properties between two bodies in contact using the slideline contact in SOL 106 and SOL 129. SOL 400 is the recommended approach for contact analysis.

**Format:**

| 1     | 2   | 3 | 4     | 5   | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|-------|-----|---|---|---|---|----|
| BFRIC | FID |   | FSTIF | MU1 |   |   |   |   |    |

**Example:**

|       |    |  |  |     |  |  |  |  |  |
|-------|----|--|--|-----|--|--|--|--|--|
| BFRIC | 33 |  |  | 0.3 |  |  |  |  |  |
|-------|----|--|--|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| FID        | Friction identification number. See Remark 1. (Integer > 0)  |
| FSTIF      | Frictional stiffness in stick. See Remarks 2. and 3. (Real > 0.0; Default = automatically selected by the program) |
| MU1        | Coefficient of static friction. (Real > 0.0)   |

**Remarks:**

1. This identification number must be unique with respect to all other friction identification numbers. This is used in the FRICID field of BCONP Bulk Data entry.
2. The value of frictional stiffness requires care. A method of choosing its value is to divide the expected frictional strength ( $MU1 * \text{expected normal force}$ ) by a reasonable value of the relative displacement that may be allowed before slip occurs. The relative value of displacement before slip occurs must be small compared to expected relative displacements during slip. A large stiffness value may cause poor convergence, while too small a value may cause poor accuracy.  
Frictional stiffness specified by the user is selected as the initial value. If convergence difficulties are encountered during the analysis, the frictional stiffness may be reduced automatically to improve convergence.
3. The stiffness matrix for frictional slip is unsymmetric. However, the program does not use the true unsymmetric matrix. Instead the program uses only the symmetric terms. This is to avoid using the unsymmetric solver to reduce CPU time.

**BIAS****MESH Bias Definition**

Specifies a variation of the mesh-size in one direction for use in the MESH entry in SOL 700. The MESH entry can create a biased or non-uniform mesh. A uniform mesh consists of a number of planes separated by a fixed distance, but for a biased mesh the distance between subsequent planes can differ. The BIAS definition allows specifying the locations of planes in one direction. For a number of intervals the density of planes can be specified.

Format:

| 1    | 2   | 3       | 4  | 5    | 6    | 7 | 8 | 9 | 10 |
|------|-----|---------|----|------|------|---|---|---|----|
| BIAS | BID |         |    |      |      |   |   |   |    |
|      | X0  | GROWTH0 | N0 | DXS0 | DXE0 |   |   |   |    |
|      | X1  | GROWTH1 | N1 | DXS1 | DXE1 |   |   |   |    |
|      | Xi  | GROWTHi | Ni | DXSi | DXEi |   |   |   |    |

Example:

|      |      |     |    |     |      |  |  |  |  |
|------|------|-----|----|-----|------|--|--|--|--|
| BIAS | 100  |     |    |     |      |  |  |  |  |
|      | -4.5 | 0.2 | 15 |     |      |  |  |  |  |
|      | -1   | 1   | 20 |     |      |  |  |  |  |
|      | 1    |     |    | 0.1 | 0.46 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| BID        | Unique bias number. Must be referenced from MESH entry. (Integer; Required)  |
| Xi         | Begin coordinate of an interval. The interval ends at the $X_{i+1}$ entry. (Real; Default = 0.0)   |
| GROWTHi    | GROWTHi is the ratio between the step size at the beginning of the interval and at the end of the interval. If it is smaller than 1.0 then the mesh refines when going from the beginning of the interval to the end of the interval. (Real>0)<br>See Remarks 1.- 3. |
| Ni         | Ni is the number of elements inside the interval. (Integer>0) See Remarks 1.- 3.   |
| DXSi       | DXSi is the start element size of the interval. (Real>0) See Remarks 1.- 3.  |
| DXEi       | DXEi is the end element size of the interval. (Real>0) See Remarks 1.- 3.  |

Remarks:

1. The begin point of the first interval has to be equal to the X0 field of the MESH entry and may be left unspecified. The end point of the last interval is given by X0 + DX as specified by the MESH entry. In the example above, the first interval is given by [-4.5,-1], the second one by [-1,1] and the last one by [1,4.5], assuming that X0 + DX = 4.5 on the MESH entry that references the bias definition from IBIDX.

2. To define the bias in an interval four fields are available. These are GROWTHi, Ni, DXSi and DXEi. To specify the bias inside an interval two of these four variables have to be specified. The other two variables have to be left blank. In addition the mesh size can be chosen constant by defining Ni and leaving GROWTHi, DXSi and DXEi blank.

This gives seven methods of specifying a bias;

1. Define GROWTHi and Ni. DXSi and DXEi have to be left blank.
2. Define DXSi and DXEi. GROWTHi and Ni have to be left blank.
3. Define DXSi and Ni. GROWTHi and DXEi have to be left blank.
4. Define DXEi and Ni. GROWTHi and DXSi have to be left blank.
5. Define GROWTHi and DXSi. Ni and DXEi have to be left blank.
6. Define GROWTHi and DXEi. Ni and DXSi have to be left blank.
7. Only define Ni and leave GROWTHi, DXSi and DXEi blank.

For method 2 it can happen that the biased elements do not exactly fit into the interval.

To get a good fit a small change to defined start and end step sizes is made. These changes in general amount to a few percents.

For method 5 and 6 a small change to the specified growth factor can be made so that the biased elements fit into the interval.

3. The algorithm for each method first determines an appropriate growth factor so that the biased elements fit into the interval. In addition the number of planes is determined. For method 2, 3 and 4 this is done by using bisection. The location of the planes is given by:

$$X(i) = X(i-1) + \Delta X(i-1)$$

$$\Delta X(i) = GROWFAC \times \Delta X(i-1)$$

$$GROWTHi = GROWFAC^{N-1}$$

Here  $X(i)$  denotes the location of the Euler plane,  $\Delta X_i$  denotes the step size of the interval and  $N$  denotes the number of Euler planes. The index  $i$  runs across the Euler planes. The variable GROWFAC denotes the grow factor between planes within the interval. The locations of the planes  $X(i)$  are written to the OUT file as part of MESH output. In addition, the growth of the element sizes is written out in the next column. This is given as the ratio in element size between the layer of elements to the right of the plane and to the left of the plane. Let  $x_0, x_1$  and  $x_2$  denote three subsequent planes, then the element size to the left of the  $x_1$ -plane is given  $x_1-x_0$  and to the right it is given by  $x_2-x_1$ . The ratio by which the element size grows if one goes across the  $x_1$ -plane is:

$$\frac{x_2 - x_1}{x_1 - x_0}$$

To get physically meaningful results, this value should not exceed 1.3 or be smaller than 0.7.

Within each interval the ratio in element size equals GROWFAC. But the element size between two adjacent elements that are in two different intervals can differ from GROWFAC. Here each interval has a distinct GROWFAC variable.

For both method 2, 5 and 6 The growth factor GROWFAC and start and end step sizes that are actually used can be obtained from this plane summary in the OUT file. In this summary for each plane a growth factor and step size is specified.

Also the total number of elements is written out.

## BJOIN

Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis in SOL 700. When the failure criterion for a grid-point pair is satisfied, the grid-point pair is removed from the join and the grid-point motion is computed for the separate grid points. The join ceases to exist when all pairs of the join have failed, after which all of the grid points of the join are treated as separate grid points.

Format:

| 1     | 2   | 3   | 4   | 5    | 6     | 7  | 8 | 9  | 10 |
|-------|-----|-----|-----|------|-------|----|---|----|----|
| BJOIN | BID | SID | TOL | TYPE |       | SN |   | SS |    |
|       |     |     |     |      | MULTI |    |   |    |    |
|       |     | TF  |     |      |       |    |   |    |    |

Example:

|       |   |   |  |          |  |      |  |      |  |
|-------|---|---|--|----------|--|------|--|------|--|
| BJOIN | 1 | 2 |  | SPOTWELD |  | 1.E3 |  | 1.E3 |  |
|-------|---|---|--|----------|--|------|--|------|--|

| Descriptor | Meaning  | Type        | Default    |
|------------|--|-------------|------------|
| BID        | BJOIN number.  | Integer > 0 | Required   |
| SID        | BCGRID ID of set of grid points.   | Integer > 0 | Required   |
| TOL        | Tolerance used in matching grid point pairs.   | Real > 0.0  | 1.E-4      |
| TYPE       | Type of failure criterion.<br>SPOTWELD Spotweld-type failure.  | C           | SPOTWELD   |
| SN         | Failure force in tension.  | Real > 0.0  | No failure |
| SS         | Failure force in shear.  | Real > 0.0  | No failure |
| MULTI      | Multiple breakable joins, where the grid points must be entered as a sequence of BJOIN pairs.<br><br>YES The grid points are entered on the BCGRID entry as a sequence of BJOIN pairs.<br><br>NO The code creates BJOIN pairs for every two grid points entered on the BCGRID entry when the grid point positions fall within the tolerance (TOL). | C           | YES        |
| TF         | Failure time for nodal constraint set.   | Real        | 1.0E20     |

**Remarks:**

1. Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
2. When the failure time,  $TF$ , is reached the spot weld becomes inactive and the constrained nodes may move freely.
3. Note that the shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted.
4. For MULTI=YES, a spotweld or rivet will always be created, regardless if their distance is larger than the value of TOL. TOL will only be used for MULTI=NO.

**BLSEG****Boundary Line Segments**

Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.

Format:

| 1     | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|----|----|----|----|----|----|----|----|----|
| BLSEG | ID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

Alternate Format:

|       |    |    |        |    |      |     |  |  |  |
|-------|----|----|--------|----|------|-----|--|--|--|
| BLSEG | ID | G1 | "THRU" | G2 | "BY" | INC |  |  |  |
|-------|----|----|--------|----|------|-----|--|--|--|

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

|  |    |    |     |     |        |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|
|  | G8 | G9 | G10 | G11 | -etc.- |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|

Continuation Entry Format 2:

|  |    |        |    |      |     |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|
|  | G8 | "THRU" | G9 | "BY" | INC |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|

Example:

|       |    |      |      |    |    |    |  |  |  |
|-------|----|------|------|----|----|----|--|--|--|
| BLSEG | 15 | 5    | THRU | 21 | BY | 4  |  |  |  |
|       | 27 | 30   | 32   | 33 |    |    |  |  |  |
|       | 35 | THRU | 44   |    |    |    |  |  |  |
|       | 67 | 68   | 72   | 75 | 84 | 93 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Line segments identification number. See Remark 2. (Integer > 0)  |
| Gi         | Grid point identification numbers on a curve in a continuous topological order so that the normal to the segment points toward the other curve. See Remark 3. (Integer > 0) |
| INC        | Grid point identification number increment. See Remark 3. (Integer or blank)  |

Remarks:

1. A line segment is defined between every two consecutive grid points. Thus, the number of line segments defined is equal to the number of grid points specified minus 1. A corresponding BWIDTH Bulk Data entry may be required to define the width/thickness of each line segment. If the corresponding BWIDTH is not present, the width/thickness for each line segment is assumed to be unity.

2. ID must be unique with respect to all other BLSEG entries. Each line segment has a width in 3-D sideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width/thickness of each line segment is defined via BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWIDTH. That is, there must be a one to one correspondence between BLSEG and BWIDTH. BWIDTH Bulk Data entry may be omitted only if the width/thickness of each segment is unity.
3. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the 'CID' field of BCOMP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.
5. A curve may be closed or open. A closed curve is specified by having the last grid point identification number the same as the first grid number.
6. See BCBODY for use of BLSEG in 3D contact.

**BNDFIX**

## Fixed Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

| 1      | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|--------|-----|----|-----|----|-----|----|-----|----|----|
| BNDFIX | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

Example:

|        |   |     |    |   |  |  |  |  |  |
|--------|---|-----|----|---|--|--|--|--|--|
| BNDFIX | 2 | 135 | 14 | 6 |  |  |  |  |  |
|--------|---|-----|----|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.) |

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDFIX1**

## Fixed Boundary Degrees-of-Freedom, Alternate Form of BNDFIX Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

| 1       | 2   | 3   | 4    | 5      | 6   | 7   | 8   | 9   | 10 |
|---------|-----|-----|------|--------|-----|-----|-----|-----|----|
| BNDFIX1 | C   | ID1 | ID2  | ID3    | ID4 | ID5 | ID6 | ID7 |    |
|         | ID8 | ID9 | ID10 | -etc.- |     |     |     |     |    |

Example:

|         |     |     |    |   |    |    |    |    |  |
|---------|-----|-----|----|---|----|----|----|----|--|
| BNDFIX1 | 2   | 135 | 14 | 6 | 23 | 24 | 25 | 26 |  |
|         | 122 | 127 |    |   |    |    |    |    |  |

Alternate Format and Example:

|         |   |     |        |     |  |  |  |  |  |
|---------|---|-----|--------|-----|--|--|--|--|--|
| BNDFIX1 | C | ID1 | "THRU" | ID2 |  |  |  |  |  |
| BNDFIX1 | 3 | 6   | THRU   | 32  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| C          | Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1< ID2)   |

Remarks:

1. BNDFIX1 and BSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDFREE**

## Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

| 1       | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|---------|-----|----|-----|----|-----|----|-----|----|----|
| BNDFREE | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

Example:

|         |     |   |   |    |   |    |  |  |  |
|---------|-----|---|---|----|---|----|--|--|--|
| BNDFREE | 124 | 1 | 5 | 23 | 6 | 16 |  |  |  |
|---------|-----|---|---|----|---|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component numbers. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

Remarks:

1. BNDFREE and CSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDFRE1**

## Free Boundary Degrees-of-Freedom, Alternate Form of BNDFREE Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

**Format:**

| 1       | 2   | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|-----|-----|--------|-----|-----|-----|-----|-----|----|
| BNDFRE1 | C   | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|         | ID8 | ID9 | -etc.- |     |     |     |     |     |    |

**Example:**

|         |     |   |   |   |   |   |    |     |  |
|---------|-----|---|---|---|---|---|----|-----|--|
| BNDFRE1 | 124 | 1 | 5 | 7 | 6 | 9 | 12 | 122 |  |
|         | 127 |   |   |   |   |   |    |     |  |

**Alternate Formats and Examples:**

|         |   |     |        |     |  |  |  |  |  |
|---------|---|-----|--------|-----|--|--|--|--|--|
| BNDFRE1 | C | ID1 | "THRU" | ID2 |  |  |  |  |  |
| BNDFRE1 | 3 | 6   | THRU   | 32  |  |  |  |  |  |

|         |  |       |  |  |  |  |  |  |  |
|---------|--|-------|--|--|--|--|--|--|--|
| BNDFRE1 |  | "ALL" |  |  |  |  |  |  |  |
| BNDFRE1 |  | ALL   |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks) |
| IDi        | Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)  |
| ALL        | All a-set degrees-of-freedom will be set free (included in c-set).   |

**Remarks:**

1. BNDFRE1 and CSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDGRID****Boundary Grid Points**

Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).

**Format:**

| 1       | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| BNDGRID | C   | GP1    | GP2 | GP3 | GP4 | GP5 | GP6 | GP7 |    |
|         | GP8 | -etc.- |     |     |     |     |     |     |    |

**Example:**

|         |     |    |    |    |    |    |    |    |  |
|---------|-----|----|----|----|----|----|----|----|--|
| BNDGRID | 123 | 41 | 42 | 43 | 44 | 45 | 46 | 47 |  |
|         | 49  |    |    |    |    |    |    |    |  |

**Alternate Format and Example:**

|         |     |     |        |     |  |  |  |  |  |
|---------|-----|-----|--------|-----|--|--|--|--|--|
| BNDGRID | C   | GP1 | "THRU" | GP2 |  |  |  |  |  |
| BNDGRID | 123 | 41  | THRU   | 49  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1. |
| GPi        | Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)     |

**Remarks:**

1. C specifies the components for the listed grid points for which boundary motion is prescribed.
2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.
3. Both fixed and free shape boundary grid point identification numbers are listed on this entry.
4. The degrees-of-freedom specified on BNDGRID entries must be sufficient to statically constrain the model.
5. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

**BOLT**

Defines the Multi-Point Constraints for a Bolt

Defines a rigid bolt by a set of MPC constraints.

**Format:**

| 1    | 2      | 3     | 4   | 5    | 6   | 7   | 8   | 9   | 10 |
|------|--------|-------|-----|------|-----|-----|-----|-----|----|
| BOLT | ID     | GRIDC |     |      |     |     |     |     |    |
|      | TOP    | GT1   | GT2 | GT3  | GT4 | GT5 | GT6 | GT7 |    |
|      |        | GT8   | GT9 | etc. |     |     |     |     |    |
|      | BOTTOM | GB1   | GB2 | GB3  | GB4 | GB5 | GB6 | GB7 |    |
|      |        | GB8   | GB9 | etc. |     |     |     |     |    |

**Example:**

|      |        |      |     |     |     |     |  |  |  |
|------|--------|------|-----|-----|-----|-----|--|--|--|
| BOLT | 100    | 1025 |     |     |     |     |  |  |  |
|      | TOP    | 101  | 102 | 103 | 104 | 105 |  |  |  |
|      | BOTTOM | 1    | 2   | 3   | 4   | 5   |  |  |  |

| Descriptor     | Meaning   |
|----------------|---|
| ID             | Element ID of the bolt. (Integer; Required; no Default)   |
| GRIDC          | Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)  |
| TOP            | Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)                                       |
| GT1, GT2, etc. | Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)  |
| BOTTOM         | Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default) |
| GB1, GB2, etc. | Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)   |

**Remarks:**

1. The GRIDS entries of the TOP and BOTTOM keywords are open-ended.
2. GRIDC is the control grid point and usually not connected to any element.
3. (GT<sub>i</sub>, GB<sub>i</sub>) are pairs of grid on top and bottom.
4. To each pair of (GT<sub>i</sub>, GB<sub>i</sub>) and GRIDC, MPCs are created internally to all 6 DOFs. Since the GBs always belong to dependent-DOFs, they cannot be applied to any SPC, SPC1, SPCD and SPCR.
5. Same number of grid points in TOP and BOTTOM. They should be coincident but it is not required. Users who do otherwise do so at their own risk since the current design does not consider the initial offset between them.

6. Bolt loads, including enforced motion, are usually prescribed on GRIDC to represent the pre-tension, overlap or loading of the bolt. BOLT relative displacements are given in the global coordinate system of the control node.
7. Global Coordinate System may have to be defined at the Control Node if the bolt direction is not a Basic Coordinate direction and the user wants to apply the loads along the shaft of the bolt.
8. Loads in directions other than the shaft of the bolt direction are possible.
9. The internally written MPC relationship is of the form:

$$u_{GB} = u_{GT} + u_{GC}$$

10. In 3D Contact analysis, it replaces GBi (Bottom bolt segment) by GTi (Top bolt segment) on the internally generated contact surface, which makes contact surface continuous across the mesh split between them.
11. Force output is obtained through Case Control MPCFORCE.
12. The internal MPCs generated by the BOLT entry are valid for small rotations only.
13. The BOLT entry must be defined in the residual structure only.
14. Sufficient boundary conditions must be placed on the BOLT to prevent mechanisms.
15. In contact analysis, care should be taken that the preload displacement does not exceed the local element width.
16. Pretension of a BOLT is a three step process in SOL 400:

#### SUBCASE 1

##### STEP 1

- LOAD=n      Applies a pretension load to the bolt on the control grid.  
 SPC=m      If any required to prevent motion of structure do not include the control grid.

##### STEP 2

- LOAD=p1    Load entry pointing to a SPCR (relative value of 0.0) and any other desired type of loading.  
 SPC=q1    SPC or SPC1 includes control grid and any other desired SPC requirements.

##### STEP 3

- LOAD=p2    Load entry pointing to a SPCR and any other additional desired type of loading.  
 SPC=q2    SPC or SPC1 includes control grid and any other additional desired SPC requirements.

17. The BOLT may also be used in SOL101 and SOL103. In this case PARAM, AUTOMSET, YES is recommended.

## BOUTPUT

### Output for Slideline Contact

Defines slave nodes at which output is requested.

Format:

| 1       | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|----|----|----|----|----|----|----|----|
| BOUTPUT | ID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

Alternate Format:

|         |    |    |        |    |      |     |  |  |  |
|---------|----|----|--------|----|------|-----|--|--|--|
| BOUTPUT | ID | G1 | "THRU" | G2 | "BY" | INC |  |  |  |
|---------|----|----|--------|----|------|-----|--|--|--|

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

|  |    |    |     |     |        |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|
|  | G8 | G9 | G10 | G11 | -etc.- |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|

Continuation Entry Format 2:

|  |    |        |    |      |     |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|
|  | G8 | "THRU" | G9 | "BY" | INC |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|

Example:

|         |    |      |      |    |    |    |  |  |  |
|---------|----|------|------|----|----|----|--|--|--|
| BOUTPUT | 15 | 5    | THRU | 21 | BY | 4  |  |  |  |
|         | 27 | 30   | 32   | 33 |    |    |  |  |  |
|         | 35 | THRU | 44   |    |    |    |  |  |  |
|         | 67 | 68   | 72   | 75 | 84 | 93 |  |  |  |

Format and Example Using "ALL" (No continuation entry is allowed):

|         |    |     |  |  |  |  |  |  |  |
|---------|----|-----|--|--|--|--|--|--|--|
| BOUTPUT | ID | ALL |  |  |  |  |  |  |  |
| BOUTPUT | 15 | ALL |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Contact region identification number of a BCOMP entry for which output is desired, or the contact Grid ID, in 3D contact. (Integer > 0) |
| Gi         | Slave node numbers for which output is desired. (Integer > 0)   |
| INC        | Grid point identification number increment. See Remark 1. (Integer or blank)  |

**Remark:**

1. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).

**BRKSQ****Specifies Data for Brake Squeal Calculations In SOL 600**

Specifies data for brake squeal calculations in SOL 600 only. Obsolete, please see [BSQUEAL, 1439](#) for an enhanced version.

**Format:**

| 1     | 2      | 3      | 4  | 5 | 6 | 7    | 8     | 9 | 10 |
|-------|--------|--------|----|---|---|------|-------|---|----|
| BRKSQ | METH   | AVSTIF |    |   |   | GLUE | ICORD |   |    |
|       | R1     | R2     | R3 | X | Y | Z    |       |   |    |
|       | NASCMD |        |    |   |   |      |       |   |    |
|       | RCFILE |        |    |   |   |      |       |   |    |

**Example:**

|       |       |        |     |     |     |     |  |  |  |
|-------|-------|--------|-----|-----|-----|-----|--|--|--|
| BRKSQ | 1     | 5.34E6 |     |     |     |     |  |  |  |
|       | 0.0   | 0.0    | 1.0 | 2.0 | 3.0 | 4.0 |  |  |  |
|       | tran  |        |     |     |     |     |  |  |  |
|       | nastb |        |     |     |     |     |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| METH       | Method flag corresponding to the type of brake squeal calculations to be performed. (Integer; Default = 1)<br><br>0 = Performs brake squeal calculations before any nonlinear analysis has taken place (corresponds to Marc feature, 4302).<br><br>1 = Performs brake squeal calculations after all nonlinear load cases (corresponds to Marc feature, 4304).<br><br>-1 = Same as ID=0 except it corresponds to Marc feature, 4301 (not recommended).  |
| AVSTIF     | Approximate average stiffness per unit area between the pads and disk. Corresponds to Marc's PARAMETERS fifth datablock, field 1. This value is also known as the initial friction stiffness in Marc Volume C documentation. AVSTIF is used as a penalty contact stiffness for brake squeal, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default. Required field) |
| GLUE       | Flag specifying whether MPC for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC; a value of 1 means include the MPCs. See Remark 6. (Integer; Default = 1)   |

| Descriptor | Meaning  |
|------------|--|
| ICORD      | Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula Cnew=Corig+Defl where Cnew are updated coordinates, Corig are original coordinates, and Defl are the final displacements from last Marc increment. (Integer; Default = 0) |
| R1         | X direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)   |
| R2         | Y direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock   |
| R3         | Z direction cosine (basic coord system); corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)   |
| X          | X coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)  |
| Y          | Y coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)  |
| Z          | Z coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)  |
| NASCMD     | Name of a command to run Nastran (limited to 64 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. The string will be converted to lower case. See Remark 2. (Character; Default=nastran)          |
| RCFILE     | Name of an RC file to be used with a secondary Nastran job (limited to 8 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. See Remark 2. (Character; Default=nastb.rc)  |

#### Remarks:

1. This entry is used to calculate complex eigenvalues for brake squeal using unsymmetric stiffness friction matrices calculated by Marc. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
2. SOL 600 performs brake squeal calculations, using the following approach. The main (original) Nastran job with input file jid.dat or jid.bdf spawns Marc just as it does for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices that are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.

NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case.

RCFILE is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified for Linux systems. RCFI<sup>L</sup> is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.

3. MPCs are produced for contact surfaces with glued contact. DMIGs are produced for contact surfaces without glued contact. The brakes and drums should not use glued contact; other regions of the structure can use glued contact.
4. The continuation lines may be omitted if defaults are appropriate.
5. When a BRKSQ<sup>L</sup> entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
6. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPCs are output. The GLUE flag signals SOL 600 to look for these MPCs and combine them with other MPCs that might be in the model using MPCADD, or if no MPCs were originally used to add the MCPs due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPCs for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPCs with only one degree-of-freedom defined. Such MPCs will be ignored; otherwise Nastran will generate a fatal error.
7. If METH=1, a Marc t19 file will be produced.
8. The names NASCMD and RCFI<sup>L</sup> must be entered in small fixed field and start in column 9 (i.e., left justified in the field).
9. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
10. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMMARC when using the BRKSQ<sup>L</sup> entry.

**BSET**

## Fixed Analysis Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

| 1    | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|------|-----|----|-----|----|-----|----|-----|----|----|
| BSET | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

Example:

|      |   |     |    |   |  |  |  |  |  |
|------|---|-----|----|---|--|--|--|--|--|
| BSET | 2 | 135 | 14 | 6 |  |  |  |  |  |
|------|---|-----|----|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.) |

Remarks:

1. BSET and BNDFIX entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BSET1**

## Fixed Analysis Degrees-of-Freedom, Alternate Form of BSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

| 1     | 2   | 3   | 4    | 5      | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|------|--------|-----|-----|-----|-----|----|
| BSET1 | C   | ID1 | ID2  | ID3    | ID4 | ID5 | ID6 | ID7 |    |
|       | ID8 | ID9 | ID10 | -etc.- |     |     |     |     |    |

Example:

|       |     |     |    |   |    |    |    |    |  |
|-------|-----|-----|----|---|----|----|----|----|--|
| BSET1 | 2   | 135 | 14 | 6 | 23 | 24 | 25 | 26 |  |
|       | 122 | 127 |    |   |    |    |    |    |  |

Alternate Format and Example:

|       |   |     |        |     |  |  |  |  |  |
|-------|---|-----|--------|-----|--|--|--|--|--|
| BSET1 | C | ID1 | "THRU" | ID2 |  |  |  |  |  |
| BSET1 | 3 | 6   | THRU   | 32  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| C          | Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1< ID2)   |

Remarks:

1. BSET1 and BNDFIX1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BSQUEAL****Specifies Data for Brake Squeal Analysis Using SOLs 400 and 600**

Defines data for brake squeal calculations involving one wheel (Primary Format) or multiple wheels (Alternate Format) as well as other rotating or stationary sliding objects used in SOLs 400 and 600.

**Primary Format (One Wheel, SOL 400 and SOL 600):**

| 1       | 2      | 3     | 4      | 5 | 6 | 7      | 8     | 9     | 10 |
|---------|--------|-------|--------|---|---|--------|-------|-------|----|
| BSQUEAL | ID     | OMETH | AVSTIF |   |   | BSONLY | IGLUE | ICORD |    |
|         | RX     | RY    | RZ     | X | Y | Z      |       |       |    |
|         | NASCMD |       |        |   |   |        |       |       |    |
|         | RCFILE |       |        |   |   |        |       |       |    |

**Alternate Format (Multiple Wheels, SOL 600 Only):**

| 1       | 2      | 3     | 4      | 5     | 6     | 7      | 8         | 9 | 10 |
|---------|--------|-------|--------|-------|-------|--------|-----------|---|----|
| BSQUEAL | ID     | OMETH | NGROUP | ICORD | ITYPE |        |           |   |    |
|         | “BODY” | ID1   | ID2    |       |       | AVSTIF | T(AVSTIF) |   |    |
|         |        | IVEC  | IDRF1  | IDRF2 | IGLUE |        |           |   |    |
|         |        | RX1   | RY1    | RZ1   | X1    | Y1     | Z1        |   |    |
|         |        | RX2   | RY2    | RZ2   | X2    | Y2     | Z2        |   |    |
|         | “BODY” | ID1   | ID2    |       |       | AVSTIF | T(AVSTIF) |   |    |
|         |        | IVEC  | IDRF1  | IDRF2 | IGLUE |        |           |   |    |
|         |        | RX1   | RY1    | RZ1   | X1    | Y1     | Z1        |   |    |
|         |        | RX2   | RY2    | RZ2   | X2    | Y2     | Z2        |   |    |
|         | NASCMD |       |        |       |       |        |           |   |    |
|         | RCFILE |       |        |       |       |        |           |   |    |

**Example, Primary Format (SOL 400):**

| 1       | 2   | 3   | 4      | 5   | 6   | 7   | 8 | 9 | 10 |
|---------|-----|-----|--------|-----|-----|-----|---|---|----|
| BSQUEAL | 100 | 0.2 | 5.34E6 |     |     | NO  |   |   |    |
|         | 0.0 | 0.0 | 1.0    | 2.0 | 3.0 | 4.0 |   |   |    |

**Example, Primary Format (SOL 600):**

| 1       | 2        | 3   | 4      | 5   | 6   | 7   | 8 | 9 | 10 |
|---------|----------|-----|--------|-----|-----|-----|---|---|----|
| BSQUEAL |          | 1.0 | 5.34E6 |     |     |     |   |   |    |
|         | 0.0      | 0.0 | 1.0    | 2.0 | 3.0 | 4.0 |   |   |    |
|         | nastran  |     |        |     |     |     |   |   |    |
|         | nastb.rc |     |        |     |     |     |   |   |    |

## Example, Alternate Format (SOL 600):

| 1       | 2        | 3   | 4   | 5    | 6    | 7      | 8    | 9 | 10 |
|---------|----------|-----|-----|------|------|--------|------|---|----|
| BSQUEAL |          | 1.0 | 2   | 1    | 1    |        |      |   |    |
|         | BODY     | 2   | 3   |      |      | 60000. | 0    |   |    |
|         |          | 0   |     |      | 1    |        |      |   |    |
|         |          | 0.0 | 0.0 | 1.0  | 10.0 | 10.0   | 25.0 |   |    |
|         |          | 0.0 | 0.0 | -1.0 | 10.0 | 10.0   | 30.0 |   |    |
|         | BODY     | 6   | 8   |      |      | 1.0    | 5    |   |    |
|         |          | 0   | 110 | 120  | 1    |        |      |   |    |
|         |          |     |     |      |      |        |      |   |    |
|         |          |     |     |      |      |        |      |   |    |
|         | nastran  |     |     |      |      |        |      |   |    |
|         | nastb.rc |     |     |      |      |        |      |   |    |

| Descriptor               | Meaning  |   |
|--------------------------|--|---|
| ID                       | Identification number of a corresponding BSQUEAL Case Control command (Integer > 0). Ignored by SOL 600.   |   |
| OMETH                    | Specifies the corresponding load factor (or time step) where the brake squeal analysis is to be performed. (Real; Default = 0.0 in SOL 400, see Remark 5., for SOL 600 the value must be 0.0 or 1.0, Remark 11.)   |   |
| NGROUP                   | Number of pairs of contact bodies (Integer > 0; Default = 1)   |   |
| ICORD<br>(SOL 600 Only)  | Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula Cnew=Corig+Defl where Cnew are updated coordinates, Corig are original coordinates, Defl are the final displacements from last Marc increment. (Integer; Default = 0) |   |
| ITYPE<br>(SOL 600 Only)  | Flag determining type of analysis. See <a href="#">Figure 9-3</a> and <a href="#">Figure 9-4</a> (Integer; Default = 1)  |   |
|                          | 1  | Brake squeal  |
|                          | 2  | Complex eigenvalue analysis following general sliding contact analysis. |
| BSONLY<br>(SOL 400 Only) | Brake-Squeal-Only flag to control whether or not to continue nonlinear iterations after brake squeal analysis is performed. BSONLY=YES means to perform brake squeal analysis only and exit nonlinear iteration immediately; BSONLY=NO means to continue nonlinear analysis. (Character; Default = YES)                                    |   |
| “BODY”                   | Character string that signals the start of a pair of contact bodies for which instability calculations are to be made (Character; Required)  |   |
| ID1                      | BCBODY ID of the first contact body. (Integer > 0; no Default; bcbody id1 must exist)  |   |
| ID2                      | BCBODY ID of the second contact body. (Integer > 0; no Default; bcbody id2 must exist)   |   |

| Descriptor               | Meaning  |   |                   |   |   |
|--------------------------|--|---|-------------------|---|---|
| AVSTIF                   | Approximate average stiffness per unit area between the two surfaces. AVSTIF is used a penalty contact stiffness, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default; Required field)                              |   |                   |   |   |
| T(AVSTIF)                | Table ID of a TABL3Di providing temperature or spatial variation of the average stiffness (Integer; Default = 0 which means FC is constant)  |   |                   |   |   |
| IVEC                     | Flag specifying whether friction vector at center of master contact body is in the same direction as that of the slave nodes. (Integer; Default = 0) <table> <tr> <td>0</td><td>In same direction</td></tr> <tr> <td>1</td><td>Slave vector is in tangential direction</td></tr> </table>  | 0 | In same direction | 1 | Slave vector is in tangential direction |
| 0                        | In same direction  |   |                   |   |   |
| 1                        | Slave vector is in tangential direction  |   |                   |   |   |
| IDRFi<br>(SOL 600 Only)  | ID of a matching RFORCE entry (if any) for body 1 or 2. (Integer; Default = 0, which means this body is not rotating)  |   |                   |   |   |
| IGLUE<br>(SOL 600 Only)  | Flag specifying whether MPC's for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC's, a value of 1 means include the MPC's. See Remarks 12. and 14. (Integer; Default = 1)  |   |                   |   |   |
| RXi                      | X direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| RYi                      | Y direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| RZi                      | Z direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| Xi                       | X coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| Yi                       | Y coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| Zi                       | Z coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)  |   |                   |   |   |
| NASCMD<br>(SOL 600 Only) | Name of a command to run Nastran (limited to 64 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. (Character; Default = nastran) If the string exceeds 8 characters, do not enter this string using free-field also see Remark 9.        |   |                   |   |   |
| RCFILE<br>(SOL 600 Only) | Name of a RC file to be used with a secondary Nastran job (limited to 8 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. (Character; Default=nastb.rc) If the string exceeds 8 characters, do not enter this string using free-field also see Remark 9. |   |                   |   |   |

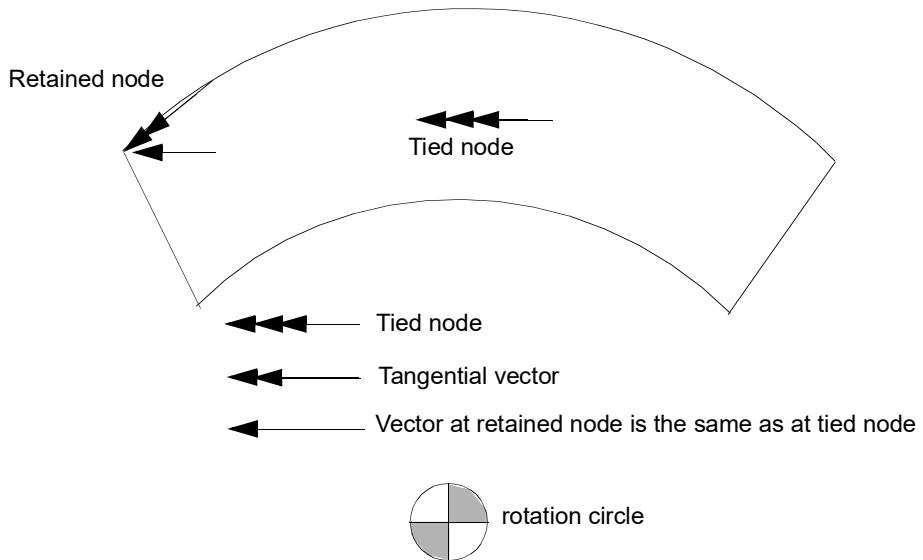


Figure 9-3

Vectors of frictional stiffness at tied and retained node.

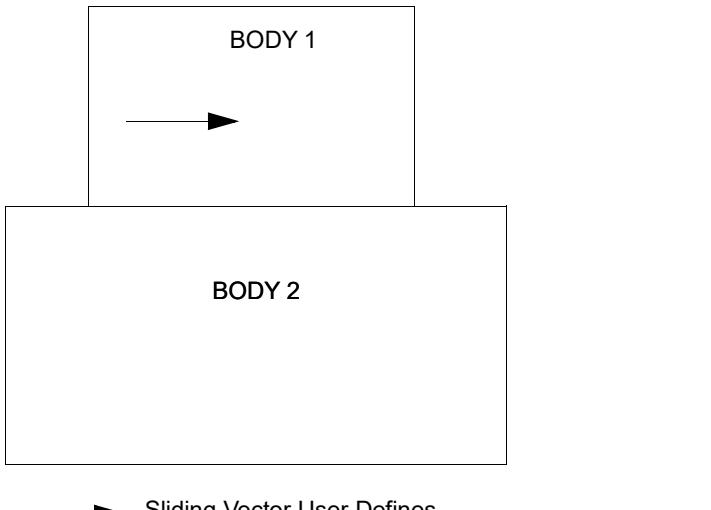


Figure 9-4

Sliding model

**Remarks:**

1. This entry is used to perform multiple-body brake squeal analysis or other contact with friction sets of bodies using unsymmetric stiffness friction matrices.

2. One or more pairs of bodies may be specified. Each pair of bodies is described by the four lines beginning with the header "BODY". For example, there may be one rotating body, such as brake disk or rotor, and one contact body or bodies, such as brake pad under each "BODY" keyword. There may be other bodies or parts in the assembly.
3. The disks should not be glued with pads. When bodies are intended to be glued, turn on BCTABLE / IGLUE or BCONPRG/IGLUE for those contact bodies.
4. Entries ICORD, ITYPE, IDBFI, IGLUE, NASCMD and RCFI are used by SOL 600 only.
5. Entry BSONLY is used by SOL 400 only.
6. Brake squeal is not available with segment-to-segment contact.

### SOL 400 Only

7. For nonlinear static analysis,  $0.0 < \text{OMETH} < 1.0$ , OMETH is overridden by the load factor from Case Control command, NLIC or its default, when a brake squeal analysis is performed in a separate SUBCASE-STEP other than ANALYSIS = NLSTATIC.
8. User subroutine UBSQUEAL is available using the BCONUDS entry.

### SOL 600 Only

9. SOL 600 performs brake squeal calculations as follows. The main (original) Nastran job with input deck jid.dat or jid.bdf spawns Marc just as done for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices which are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.  
NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case. RCFI is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified. RCFI is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.
10. OMETH can only be one of the following two values in SOL 600.  
0.0=Perform brake squeal calculations before any nonlinear analysis has taken place  
1.0=Perform brake squeal calculations after all nonlinear load cases
11. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
12. MPC's are produced for contact surfaces with glued contact. DMIG's are produced for contact surfaces without glued contact. The brakes and drums may not use glued contact, other regions of the structure can used glued contact.

13. When the BSQUEAL entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
14. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPC's are output. The GLUE flag signals SOL 600 to look for these MPC's and combine them with other MPC's that might be in the model using MPCADD, or if no MPC's were originally used to add the MCP's due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPC's for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPC's with only one degree-of-freedom defined. Such MPC's will be ignored otherwise Nastran will generate a fatal error.
15. The names NASCMD and RCFILE must be entered in small fixed field and start in column 9 (i.e. left justified in the field).
16. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
17. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMMARC when using the BSQUEAL entry.
18. The Bulk Data entry BRKSQ is the alternate format in SOL 600 when there is only one body.

**BSURF**

Contact Body or Surface

Defines a contact body or surface by Element IDs.

**Format:**

| 1     | 2     | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|-------|-------|-------|-------|-------|-------|-------|-------|-------|----|
| BSURF | ID    | ELID1 | ELID2 | ELID3 | ELID4 | ELID5 | ELID6 | ELID7 |    |
|       | ELID8 | ELID9 | etc.  |       |       |       |       |       |    |

**Alternate Format:**

|       |       |       |       |       |      |     |  |  |  |
|-------|-------|-------|-------|-------|------|-----|--|--|--|
| BSURF | ID    | ELID1 | THRU  | ELID2 | BY   | INC |  |  |  |
|       | ELID3 | THRU  | ELID4 | BY    | INC2 |     |  |  |  |

**Example:**

|       |    |      |      |    |    |    |  |  |  |
|-------|----|------|------|----|----|----|--|--|--|
| BSURF | 15 | 5    | THRU | 21 | BY | 4  |  |  |  |
|       | 27 | 30   | 32   | 33 |    |    |  |  |  |
|       | 35 | THRU | 44   |    |    |    |  |  |  |
|       | 67 | 68   | 72   | 75 | 84 | 93 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification of a deformable surface corresponding to a BSID value on the BCBODY entry. See Remark 2. (Integer > 0)   |
| ELIDI      | Element identification numbers. If the curve or surface is defined with element ids only, the direction of the normal depends on the grid point numbering. Keywords THRU and BY can be used to assist the listing. For SOL 600 and SOL 700 the Alternate Format must be used with Remarks 6. and 7. |
| INC        | Identification number increment. See Remark 3. (Integer or blank)   |

**Remarks:**

1. BSURF can be used in SOL101, 103, 105, 107, 108, 109, 110, 111, 112, 400, 600, and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. For automatic generation of element IDs, the default increment value is 1 if element numbers are increasing or -1 if element numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The deformable surface may alternately be defined using BCBOX, BCPROP, or BCMATL entries.
5. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.

6. For SOLs 600 and 700, Format 1 and the Alternate Format cannot be mixed for a particular BSURF entry.
7. For SOLs 600 and 700 if the Alternate Format is used, THRU must be in column 4 for entries with the BSURF header and in column 3 for continuation entries. If BY is not used, columns 6-9 of the BSURF entry and columns 5-9 of continuation entries must be blank.

**BWIDTH****Boundary Line Segment Width or Thickness**

Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry for SOL 106 or SOL 129. SOL 400 general contact capability, defined with BCONTACT, BCTABL1, etc. is the recommended approach for contact analysis.

**Format:**

| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|----|----|----|----|----|----|----|----|----|
| BWIDTH | ID | W1 | W2 | W3 | W4 | W5 | W6 | W7 |    |

**Alternate Format:**

|        |    |    |        |    |      |     |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|
| BWIDTH | ID | W1 | "THRU" | W2 | "BY" | INC |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|

The continuation entry formats may be used more than once and in any order. They may also be used with either format above.

**Continuation Entry Format 1:**

|  |    |    |     |     |        |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|
|  | W8 | W9 | W10 | W11 | -etc.- |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|

**Continuation Entry Format 2:**

|  |    |        |    |      |     |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|
|  | W8 | "THRU" | W9 | "BY" | INC |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|

**Example:**

|        |     |      |      |     |     |     |  |  |  |
|--------|-----|------|------|-----|-----|-----|--|--|--|
| BWIDTH | 15  | 2.0  | THRU | 5.0 | BY  | 1.0 |  |  |  |
|        | 2.0 | 2.0  | 2.0  | 2.0 |     |     |  |  |  |
|        | 35. | THRU | 44.  |     |     |     |  |  |  |
|        | 1.5 | 3.4  | 7.6  | 0.4 | 0.7 |     |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | BLSEG entry identification number. (Integer > 0)  |
| Wi         | Width values for the corresponding line segments defined in the BLSEG entry. See Remark 1. (Real > 0.0) |
| INC        | Width value increment. See Remark 2. (Real or blank)  |

**Remarks:**

1. BWIDTH may be omitted if the width of each segment defined in the BLSEG entry is unity. The number of widths to be specified is equal to the number of segments defined in the corresponding BLSEG entry.

2. The default value for INC is 1.0 if the width is increasing or -1.0 if the width is decreasing. That is, the user need not specify BY and the increment value. If the number of widths specified is less than the number of segments defined in the corresponding BLSEG entry, the width for the remaining segments is assumed to be equal to the last width specified.
3. If there is only one grid point in the corresponding BLSEG entry, there is no contributory area associated with the grid point. To compute correct contact stresses an area may be associated with the single grid point by specifying the area in field W1.

## Entries CA - CM

### CAABSF

Frequency-Dependent Acoustic Absorber Element

Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.

Format:

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|----|----|----|----|---|---|----|
| CAABSF | EID | PID | G1 | G2 | G3 | G4 |   |   |    |

Example:

|        |    |    |   |    |    |  |  |  |  |
|--------|----|----|---|----|----|--|--|--|--|
| CAABSF | 44 | 38 | 1 | 10 | 20 |  |  |  |  |
|--------|----|----|---|----|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)                               |
| PID        | Property identification number that matches a PAABSF entry. (Integer > 0; Default = EID) |
| Gi         | Grid point identification number of fluid connection points. (Integer $\geq 0$ or blank) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If only G1 is specified then a point impedance is assumed. If G1 and G2 are specified then a line impedance is assumed. If G1, G2, and G3 are specified, then an impedance is associated with the area of the triangular face. If G1 through G4 are specified, then an impedance is associated with the quadrilateral face. See [Figure 9-5](#).
3. The CAABSF element must connect entirely to fluid points on the fluid-structure boundary.
4. This element is used only in frequency response and is ignored in all other solutions.

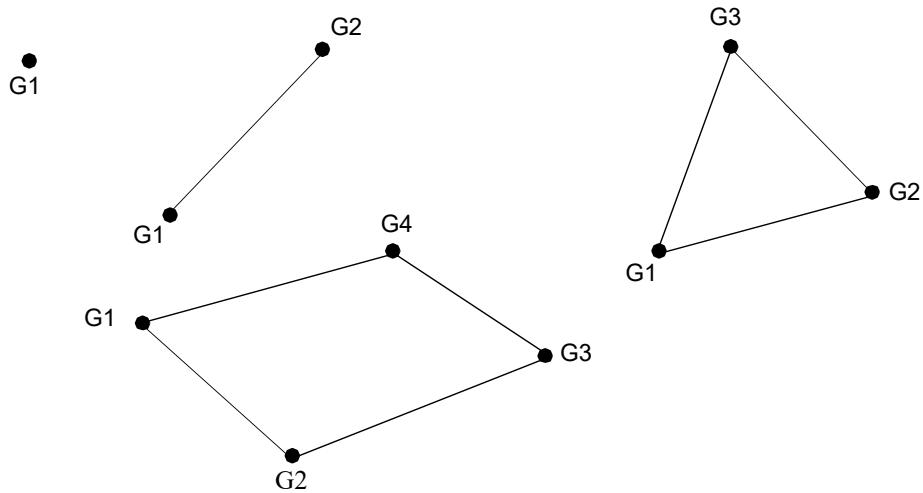


Figure 9-5 Four Types of CAABSF Elements

**CACINF3****Acoustic Conjugate Infinite Element Base Connection**

Defines an acoustic conjugate infinite element with triangular base.

Format:

| 1       | 2   | 3   | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|---------|-----|-----|----|----|----|---|---|---|----|
| CACINF3 | EID | PID | G1 | G2 | G3 |   |   |   |    |

Example:

|         |     |    |      |      |      |  |  |  |  |
|---------|-----|----|------|------|------|--|--|--|--|
| CACINF3 | 111 | 10 | 1004 | 1008 | 1011 |  |  |  |  |
|---------|-----|----|------|------|------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element Identification Number. (0 < Integer < 100,000,000)                         |
| PID        | Property Identification Number of a PACINF entry. (Integer > 0)                    |
| Gi         | Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.

**CACINF4**

## Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with quadrilateral base.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|-----|-----|----|----|----|----|---|---|----|
| CACINF4 | EID | PID | G1 | G2 | G3 | G4 |   |   |    |

**Example:**

|         |     |    |      |      |      |      |  |  |  |
|---------|-----|----|------|------|------|------|--|--|--|
| CACINF4 | 275 | 10 | 1027 | 1032 | 1056 | 1021 |  |  |  |
|---------|-----|----|------|------|------|------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element Identification Number. (0 < Integer < 100,000,000)                         |
| PID        | Property Identification Number of a PACINF entry. (Integer > 0)                    |
| Gi         | Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.

## CAERO1

## Aerodynamic Panel Element Connection

Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory for subsonic aerodynamics and the ZONA51 theory for supersonic aerodynamics.

**Format:**

| 1      | 2   | 3   | 4  | 5     | 6      | 7     | 8      | 9    | 10 |
|--------|-----|-----|----|-------|--------|-------|--------|------|----|
| CAERO1 | EID | PID | CP | NSPAN | NCHORD | LSPAN | LCHORD | IGID |    |
|        | X1  | Y1  | Z1 | X12   | X4     | Y4    | Z4     | X43  |    |

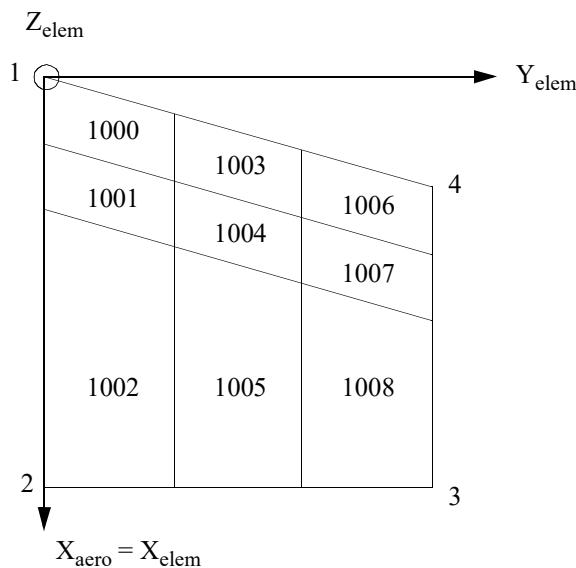
**Example:**

|        |      |     |     |     |     |     |     |     |  |
|--------|------|-----|-----|-----|-----|-----|-----|-----|--|
| CAERO1 | 1000 | 1   |     | 3   |     |     | 2   | 1   |  |
|        | 0.0  | 0.0 | 0.0 | 1.0 | 0.2 | 1.0 | 0.0 | 0.8 |  |

| Descriptor               | Meaning   |
|--------------------------|---|
| EID                      | Element identification number. (0 < Integer < 100,000,000)  |
| PID                      | Property identification number of a PAERO1 entry; used to specify associated bodies. Required even if there are no associated bodies. (Integer > 0)                                       |
| CP                       | Coordinate system for locating points 1 and 4. (Integer $\geq 0$ ; Default = 0)   |
| NSPAN                    | Number of spanwise boxes; if a positive value is given NSPAN, equal divisions are assumed; if zero or blank, a list of division points is given at LSPAN, field 7. (Integer $\geq 0$ )    |
| NCHORD                   | Number of chordwise boxes; if a positive value is given NCHORD, equal divisions are assumed; if zero or blank, a list of division points is given at LCHORD, field 8. (Integer $\geq 0$ ) |
| LSPAN                    | ID of an AEFACT entry containing a list of division points for spanwise boxes. Used only if NSPAN, field 5 is zero or blank. (Integer > 0)  |
| LCHORD                   | ID of an AEFACT data entry containing a list of division points for chordwise boxes. Used only if NCHORD, field 6 is zero or blank. (Integer > 0)   |
| IGID                     | Interference group identification; aerodynamic elements with different IGIDs are uncoupled. (Integer > 0)   |
| X1, Y1, Z1<br>X4, Y4, Z4 | Location of points 1 and 4, in coordinate system CP. (Real)<br>}  |
| X12, X43                 | Edge chord lengths in aerodynamic coordinate system. (Real $\geq 0.0$ , but not both zero.)   |

**Remarks:**

1. The boxes and corner point nodes are numbered sequentially, beginning with EID. The user should be careful to ensure that all box and corner point node numbers are unique. There can be overlapping IDs between the structural and aerodynamic model, but MSC Patran will not then be able to display any results. Also, non-unique corner IDs are allowed, but results cannot be visualized in MSC Patran.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0. and 1. (In which case the corners of the panel would not be at the reference points.)
3. A triangular element is formed if X12 or X43=0.0
4. The element coordinate system is right-handed as shown in [Figure 9-6](#).
5. The continuation is required.
6. It is recommended that NCHORD or LCHORD be chosen so that the typical box chord length  $\Delta x$  satisfies the condition  $\Delta x < 0.08 V/f$  (recent studies indicate that .02  $V/f$  is needed to get converged stability derivatives) where  $V$  is the minimum velocity and  $f$ , in hertz, is the maximum frequency to be analyzed (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

**Figure 9-6** Element Coordinate System for Aerodynamic Panel

7. This entry can be used for two different aerodynamic theories: Doublet-Lattice for subsonic and ZONA51 for supersonic. The proper theory is selected based on the specification of Mach number on the MKAEROi or TRIM entry.

**CAERO2****Aerodynamic Body Connection**

Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

**Format:**

| 1      | 2   | 3   | 4  | 5   | 6    | 7   | 8    | 9    | 10 |
|--------|-----|-----|----|-----|------|-----|------|------|----|
| CAERO2 | EID | PID | CP | NSB | NINT | LSB | LINT | IGID |    |
|        | X1  | Y1  | Z1 | X12 |      |     |      |      |    |

**Example:**

|        |      |      |      |      |   |    |  |   |  |
|--------|------|------|------|------|---|----|--|---|--|
| CAERO2 | 1500 | 2    | 100  |      | 4 | 99 |  | 1 |  |
|        | -1.0 | 100. | -30. | 175. |   |    |  |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )   |
| PID        | Property identification number of a PAERO2 entry. ( $\text{Integer} > 0$ )  |
| CP         | Coordinate system for locating point 1. ( $\text{Integer} \geq 0$ ; Default = 0)  |
| NSB        | Number of slender body elements. If NSB > 0, then NSB equal divisions are assumed; if zero or blank, specify a list of divisions in LSB. ( $\text{Integer} \geq 0$ )    |
| NINT       | Number of interference elements. If NINT > 0, then NINT equal divisions are assumed; if zero or blank, specify a list of divisions in LINT. ( $\text{Integer} \geq 0$ ) |
| LSB        | ID of an AEFACT Bulk Data entry for slender body division points; used only if NSB is zero or blank. ( $\text{Integer} \geq 0$ )  |
| LINT       | ID of an AEFACT data entry containing a list of division points for interference elements; used only if NINT is zero or blank. ( $\text{Integer} \geq 0$ )              |
| IGID       | Interference group identification. Aerodynamic elements with different IGIDs are uncoupled. ( $\text{Integer} \geq 0$ )   |
| X1, Y1, Z1 | Location of point 1 in coordinate system CP. (Real)   |
| X12        | Length of body in the x-direction of the aerodynamic coordinate system. (Real $> 0.0$ )   |

**Remarks:**

1. Point 1 is the leading point of the body.
2. All CAERO1 (panels) and CAERO2 (bodies) in the same group (IGID) will have aerodynamic interaction.
3. At least one interference element is required for the aerodynamic body specified by this entry.

4. The beams and connection points are numbered sequentially beginning with EID. The user should be careful to ensure that all aero elements and connection point IDs are unique. Overlapping IDs between structure and aerodynamic models are allowed, but will prevent results visualization in Patran.

Old rules regarding numbering among Z, ZY, Y bodies and CAERO1 no longer apply: arbitrary ordering is allowed.

5. At least two slender body elements are required for each aerodynamic body.
6. Interference elements are only intended for use with panels.
7. Determining the size of the j-set (i.e., the number of aerodynamic elements) is essential to input D1JE and D2JE matrices. Use the following expressions for locating the proper row in the two matrices:

$$\begin{aligned} J = & \text{Number of boxes} + \text{Number of I-elements, z} \\ & + 2 * (\text{Number of I-elements, zy}) \\ & + \text{Number of I-elements, y} \\ & + \text{Number of S-elements, z} \\ & + 2 * (\text{Number of S-elements, zy}) \\ & + \text{Number of S-elements, y} \end{aligned}$$

where I-elements denote interference and S-elements denote slender body.

## CAERO3

## Aerodynamic Panel Element Configuration

Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.

Format:

| 1      | 2   | 3   | 4  | 5     | 6      | 7      | 8  | 9   | 10 |
|--------|-----|-----|----|-------|--------|--------|----|-----|----|
| CAERO3 | EID | PID | CP | LISTW | LISTC1 | LISTC2 |    |     |    |
|        | X1  | Y1  | Z1 | X12   | X4     | Y4     | Z4 | X43 |    |

Example:

|        |      |      |     |      |     |      |    |      |  |
|--------|------|------|-----|------|-----|------|----|------|--|
| CAERO3 | 2000 | 2001 | 0   | 22   | 33  |      |    |      |  |
|        | 1.0  | 0.0  | 0.0 | 100. | 17. | 130. | 0. | 100. |  |

| Descriptor               | Meaning  |
|--------------------------|--|
| EID                      | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )  |
| PID                      | Property identification number of a PAERO3 entry. ( $\text{Integer} > 0$ )   |
| CP                       | Coordinate system for locating points 1 and 4. ( $\text{Integer} \geq 0$ ; Default = 0)  |
| LISTW                    | Identification number of an AEFACT entry that lists (x,y) pairs for structural interpolation of the wing. ( $\text{Integer} > 0$ ) |
| LISTC1,<br>LISTC2        | Identification number of AEFACT entries that list (x,y) pairs for control surfaces, if they exist. ( $\text{Integer} \geq 0$ )     |
| X1, Y1, Z1<br>X4, Y4, Z4 | Location of points 1 and 4 in coordinate system CP. (Real)   |
| X12, X43                 | Edge chord lengths in the aerodynamic coordinate system. ( $\text{Real} \geq 0$ , $X12 \neq 0$ )                                   |

Remarks:

1. EID must be unique with respect to all other element identification numbers.
2. The (x,y) pairs on LISTW, LISTC1 and LISTC2 AEFACT entries are in the aero element coordinate system (see [Figure 9-7](#)). The (x,y) pairs define a set of aerodynamic grid points that are independent of Mach number and are selected by the user to be representative of the planform and motions of interest. The (x,y) pairs must be sufficient in number and distribution such that: the surface spline provides an accurate interpolation between them and the Mach Box centers that are variously located on the planform as a function of Mach number (a complete description of the Mach Box Method is given in the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

3. The (x,y) pairs are numbered sequentially, beginning with EID for LISTW, then LISTC1, and finally for LISTC2. On SPLINEi entries, the box numbers (BOX1 and BOX2 on SPLINE1, ID1 and ID2 on SPLINE2, and UKID on SPLINE3) refer to the (x,y) pair sequence number appropriate for the surface (primary, or first or second control) being splined.
4. If cranks and/or control surfaces exist, their locations are given on the PAERO3 entry.
5. The numbering system and coordinate system are shown below:

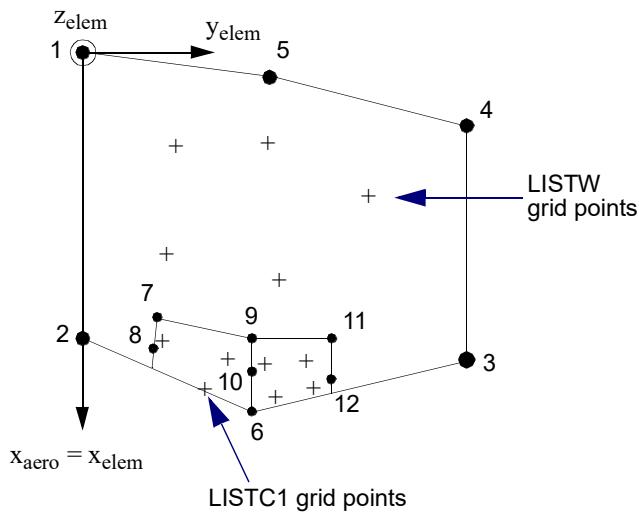


Figure 9-7 CAERO3 Element Configuration

| Planform Corners |                         | Control  |
|------------------|-------------------------|--|
| 1                | Leading edge, inboard   | 7 Hinge line, inboard                          |
| 2                | Trailing edge, inboard  | 8 On inboard edge (usually at trailing edge)   |
| 3                | Trailing edge, outboard | 9 Hinge line, outboard                         |
| 4                | Leading edge, outboard  | 10 On outboard edge (usually at trailing edge) |
| Crank            |                         | Control (if two)                               |
| 5                | Leading edge            | 9 Hinge line, inboard                          |
| 6                | Trailing edge           | 10 On inboard edge (usually at trailing edge)  |
|                  |                         | 11 Hinge line, outboard                        |
|                  |                         | 12 On outboard edge (usually at trailing edge) |

6. The CAERO3 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO3.

**CAERO4****Aerodynamic Macro-Strip Element Connection**

Defines an aerodynamic macro element for Strip theory.

**Format:**

| 1      | 2   | 3   | 4  | 5     | 6     | 7  | 8  | 9   | 10 |
|--------|-----|-----|----|-------|-------|----|----|-----|----|
| CAERO4 | EID | PID | CP | NSPAN | LSPAN |    |    |     |    |
|        | X1  | Y1  | Z1 | X12   | X4    | Y4 | Z4 | X43 |    |

**Example:**

|        |      |      |     |     |     |     |     |     |  |
|--------|------|------|-----|-----|-----|-----|-----|-----|--|
| CAERO4 | 6000 | 6001 | 100 |     | 315 |     |     |     |  |
|        | 0.0  | 0.0  | 0.0 | 1.0 | 0.2 | 1.0 | 0.0 | 0.8 |  |

| Descriptor               | Meaning  |
|--------------------------|--|
| EID                      | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )  |
| PID                      | Property identification number of a PAERO4 entry. ( $\text{Integer} > 0$ )   |
| CP                       | Coordinate system for locating points 1 and 4. ( $\text{Integer} \geq 0$ ; Default = 0)  |
| NSPAN                    | Number of strips; if a positive value is given, NSPAN equal strips are assumed. If zero or blank, LSPAN must be specified. ( $\text{Integer} \geq 0$ ) |
| LSPAN                    | ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. ( $\text{Integer} > 0$ )                   |
| X1, Y1, Z1<br>X4, Y4, Z4 | Location of points 1 and 4 in coordinate system CP. (Real)<br>}  |
| X12, X43                 | Edge chord lengths in aerodynamic coordinate system. (Real $\geq 0.0$ , and not both zero.)  |

**Remarks:**

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and greater than structural grid, scalar, and extra point IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN = 3, the division points are 0.0, 0.333, 0.667, and 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.

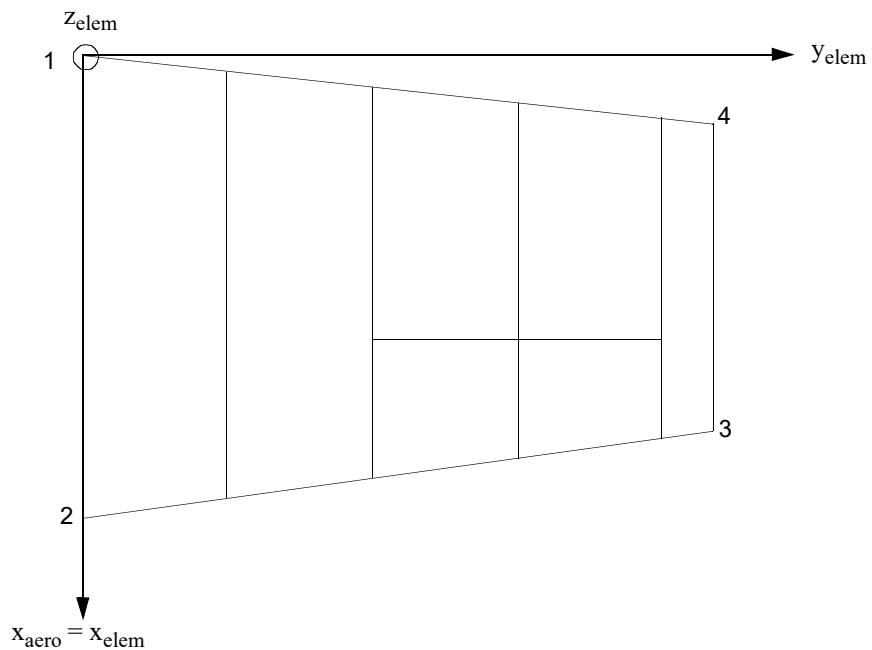


Figure 9-8      CAERO4 Element Connection

4. The CAERO4 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO4.

## CAERO5

## Aerodynamic Panel Element Configuration

Defines an aerodynamic macro element for Piston theory.

Format:

| 1      | 2   | 3   | 4  | 5     | 6     | 7     | 8      | 9   | 10 |
|--------|-----|-----|----|-------|-------|-------|--------|-----|----|
| CAERO5 | EID | PID | CP | NSPAN | LSPAN | NTHRY | NTHICK |     |    |
|        | X1  | Y1  | Z1 | X12   | X4    | Y4    | Z4     | X43 |    |

Example:

|        |      |      |     |     |     |     |    |     |  |
|--------|------|------|-----|-----|-----|-----|----|-----|--|
| CAERO5 | 6000 | 6001 | 100 |     | 315 | 0   | 0  |     |  |
|        | 0.0  | 0.0  | 0.0 | 1.0 | 0.2 | 1.0 | 0. | 0.8 |  |

| Descriptor             | Meaning  |  |
|------------------------|--|--|
| EID                    | Element identification number. (0 < Integer < 100,000,000)   |  |
| PID                    | Property identification number of a PAERO5 entry. (Integer > 0)  |  |
| CP                     | Coordinate system for locating points 1 and 4. (Integer $\geq 0$ ; Default = 0)  |  |
| NSPAN                  | Number of strips. (Integer. If a positive value is given, equal strips are assumed. If zero or blank, then LSPAN must be specified.) |  |
| LSPAN                  | ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)            |  |
| NTHRY                  | Parameter to select Piston or van Dyke's theory. (Integer = 0, 1, or 2; Default = 0)   |  |
|                        | Blank or 0   | Piston theory is used to compute $\bar{C}_1$ and $\bar{C}_2$   |
|                        | 1  | van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with no sweep correction ( $\sec\Lambda = 1.0$ ).       |
|                        | 2  | van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with a sweep correction based on the actual $\Lambda$ . |
| NTHICK                 | Parameter to select thickness integrals input. (Integer $\geq 0$ ; Default = 0)  |  |
|                        | Blank or 0   | Thickness integrals are computed internally.   |
|                        | > 0  | Thickness integrals are input directly and are the ID number of an AEFACT entry that lists the $I$ and/or $J$ integrals. |
| X1,Y1, Z1<br>X4,Y4, Z4 | {}   | Location of points 1 and 4 in coordinate system CP. (Real)   |
| X12, X43               |  | Edge chord lengths in aerodynamic coordinate system. (Real $\geq 0$ ; X12 and X43 cannot both be zero.)                  |

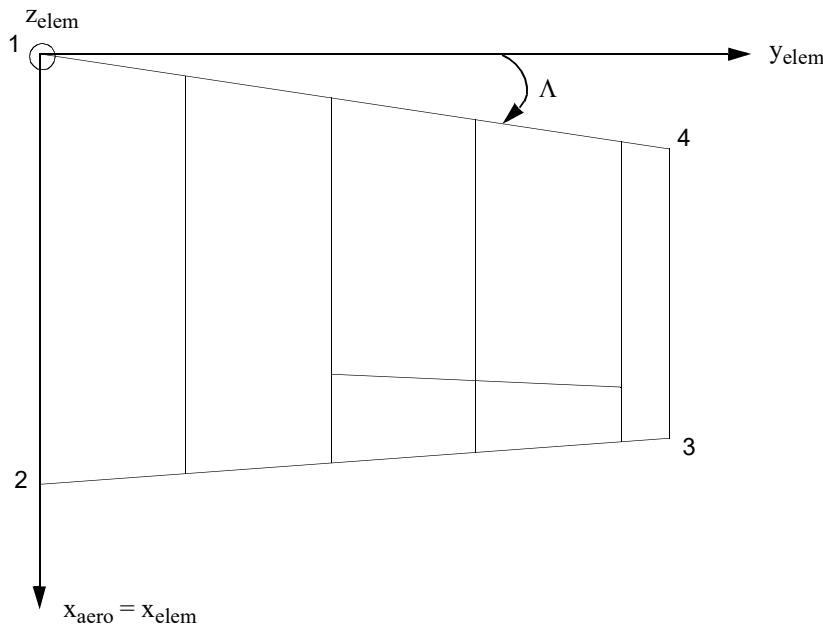


Figure 9-9 CAERO5 Element Configuration

**Remarks:**

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and different from structural grid IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if  $X12$  or  $X43 = 0.0$ .

4. 
$$\bar{C}_1 = m / (m^2 - \sec^2 \Lambda)^{1/2}$$
- $$\bar{C}_2 = [m^4(\gamma + 1) - 4 \sec^2 \Lambda (m^2 - \sec^2 \Lambda)] / [4(m^2 - \sec^2 \Lambda)^2]$$

where:

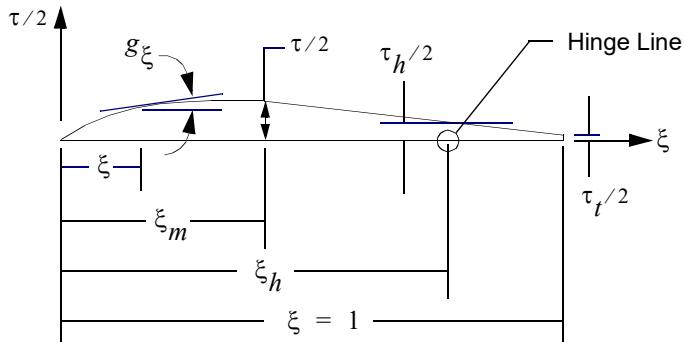
$m$  = Mach number

$\gamma$  = Specific heat ratio

$\Lambda$  = Leading edge sweep angle

- When  $\sec \Lambda = 0.0$ , Piston theory coefficients are obtained (NTHRY = 1)  
 When  $\sec \Lambda = 1.0$ , van Dyke's coefficients are obtained (NTHRY = blank or 0)  
 When  $\sec \Lambda \neq 0.0$  or  $\neq 1.0$ , sweep corrections are included (NTHRY = 2)

5.  $I$  and  $J$  thickness integral definitions:



$$g_\xi \equiv \frac{dg}{d\xi} = \text{slope of airfoil semithickness}$$

$$I_1 = \int_0^1 g_\xi d\xi \quad J_1 = \int_{\xi_h}^1 g_\xi d\xi$$

$$I_2 = \int_0^1 \xi g_\xi d\xi \quad J_2 = \int_{\xi_h}^1 \xi g_\xi d\xi$$

$$I_3 = \int_0^1 \xi^2 g_\xi d\xi \quad J_3 = \int_{\xi_h}^1 \xi^2 g_\xi d\xi$$

$$I_4 = \int_0^1 g_\xi^2 d\xi \quad J_4 = \int_{\xi_h}^1 g_\xi^2 d\xi$$

$$I_5 = \int_0^1 \xi g_\xi^2 d\xi \quad J_5 = \int_{\xi_h}^1 \xi g_\xi^2 d\xi$$

$$I_6 = \int_0^1 \xi^2 g_\xi^2 d\xi \quad J_6 = \int_{\xi_h}^1 \xi^2 g_\xi^2 d\xi$$

**Figure 9-10    CAERO5 / and J Thickness Integral Definitions**

6. The CAERO5 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO5.

**CAMPBLL****Campbell Diagram Parameters**

Specifies the parameters for Campbell Diagram generation and mode tracking analysis.

Formats:

| 1       | 2      | 3     | 4       | 5      | 6      | 7 | 8 | 9 | 10 |
|---------|--------|-------|---------|--------|--------|---|---|---|----|
| CAMPBLL | CID    | VPARM | DDVALID | TYPE   |        |   |   |   |    |
|         | MODTRK | CORU  | SWITR   | NUMMOD | PRTCOR |   |   |   |    |

Examples:

|         |    |       |    |     |   |  |  |  |  |
|---------|----|-------|----|-----|---|--|--|--|--|
| CAMPBLL | 15 | SPEED | 22 | RPM |   |  |  |  |  |
|         | 1  | 0.75  | 1  |     | 1 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| CID        | Identification number of entry (Integer > 0; Required).   |
| VPARM      | Variable parameter, allowable entry is: 'SPEED'.<br>'SPEED', reference rotor speed will be varied (rotordynamic option only).       |
| DDVALID    | Identification number of DDVAL entry that specifies the values for the variable parameter (Integer > 0; Required).                  |
| TYPE       | Allowable entries are: 'FREQ' and 'RPM'.  |
| MODTRK     | Perform mode tracking using either numerical approach (default) or eigenvector based approach or both. (Default = 0). See remark 4. |
| CORU       | Threshold for mode correlation.   |
| SWITR      | Option to use updated mode for mode tracking in case of failure. See remark 7.  |
| NUMMODE    | Number of modes to track (Should be less than 0.5*Neig, where Neig is the number of eigenvalues extracted in EIGC). See remark 4.   |
| PRTCOR     | Option to print correlation matrix at each rotor speed (Default = 0). See remark 9.   |

Remarks:

1. CAMPBLL option is supported for both SOL 107 and SOL 110 analysis.
2. CAMPBLL option is not supported in SOL 200 and SOL 400 for ANALYSIS = DCEIG/MCEIG.
3. Only the modes with positive imaginary part are used for mode tracking (to avoid repeated eigenvalues.)
4. Following methods are available in MSC Nastran for mode tracking:
  - a. Numerical Mode tracking based on second derivative.  
Obtain second derivative for the variation of eigenvalue with rotor speed and determine the eigenfrequency with minimum slope.

- b. Mode tracking based on orthogonality of left and right eigenvectors.

This feature is available for analysis in fixed reference frame only.

MODTRK = 0: Perform numerical mode tracking only. (Default)

MODTRK = 1: Perform BOTH numerical and eigenvector based mode tracking.

MODTRK = 2: Perform eigenvector based mode tracking only.

5. The variables specified in the second row are relevant for eigenvector-based mode tracking only.
6. Ideally, for tracked pair of modes, the value of Normalized Cross Complex Orthogonality (NC2O) parameter is very close to 1. However, for complex models involving large stator component and dynamic reduction, this value can be significantly lower. Thus, the default value for CORU is set to 0.7. Based on the model complexity and NC2O matrix obtained for two rotor speeds, user may change this value.
7. In case of mode tracking failure at a particular rotor speed, user has the option to continue mode tracking for rest of the rotors speed by:
  - SWITR = 1: Updating modes corresponding to rotor speed with failed mode tracking, or
  - SWITR = 0: Using the modes from last rotor speed with successful mode tracking. (Default)
 In case the Campbell diagram includes multiple mode switching and there are new modes entering the analysis, use of SWITR=1 is recommended.
8. Only the complex eigenvalues with positive imaginary part are considered for mode tracking analysis. As a result, the number of eigenvalues available for mode tracking may not be same at all the rotor speeds considered in the analysis. This may lead to mode tracking failure. One way to avoid this issue is to pick NUMMODE to be less than half of the number of complex eigenvalues extracted using EIGC bulk data entry. Thus, the analysis uses only the first NUMMODE frequencies with positive imaginary part for mode tracking at each rotor speed.
9. When PRTCOR parameter is turned to 1, the code prints NC2O matrix corresponding to each rotor speed in the F06 file. This information can be very useful in identifying reasons for mode tracking failure.
10. For both approaches, damping ratio is also tracked while tracking eigenfrequencies. The results produced from mode tracking can be extracted in OP4 using following assign statements:

```
$ Extract results for numerical mode tracking
ASSIGN OUTPUT4='freq_nmt.op4',UNIT=71,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_nmt.op4',UNIT=72,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_nmt.op4',UNIT=73,FORM=FORMATTED,DELETE $ DAMPING RATIO
$ Extract results for eigenvector based mode tracking
ASSIGN OUTPUT4='freq_evmt.op4',UNIT=81,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_evmt.op4',UNIT=82,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_evmt.op4',UNIT=83,FORM=FORMATTED,DELETE $ DAMPING RATIO
```

These OP4 files can be imported in MSC PATRAN to produce Campbell Diagram.

11. DMP>1 is not supported for SOL 110 for generating Campbell diagram when ACMS option is not used.

**CAXIFi****Fluid Element Connections**

Defines an axisymmetric fluid element that connects i = 2, 3, or 4 fluid points.

**Formats:**

| 1      | 2   | 3    | 4    | 5    | 6    | 7   | 8 | 9 | 10 |
|--------|-----|------|------|------|------|-----|---|---|----|
| CAXIF2 | EID | IDF1 | IDF2 |      |      | RHO | B |   |    |
| CAXIF3 | EID | IDF1 | IDF2 | IDF3 |      | RHO | B |   |    |
| CAXIF4 | EID | IDF1 | IDF2 | IDF3 | IDF4 | RHO | B |   |    |

**Examples:**

|        |     |     |     |     |     |         |       |  |  |
|--------|-----|-----|-----|-----|-----|---------|-------|--|--|
| CAXIF2 | 11  | 23  | 25  |     |     | 0.25E-3 |       |  |  |
| CAXIF3 | 105 | 31  | 32  | 33  |     | 6.47E-3 |       |  |  |
| CAXIF4 | 524 | 421 | 425 | 424 | 422 | 0.5E-3  | 2.5+3 |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)      |
| IDFi       | Identification numbers of connected GRIDF points. (Integer > 0) |
| RHO        | Fluid density in mass units. (Real > 0.0 or blank)              |
| B          | Fluid bulk modulus. (Real $\geq$ 0.0 or blank)                  |

**Remarks:**

1. CAXIFi is allowed only if an AXSLOT entry is also present.
2. The element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO or B is blank, then the corresponding RHOD and BD fields must be specified on the AXSLOT entry.
4. Plot elements are generated for these elements. Because each plot element connects two points, one is generated for the CAXIF2 element, three are generated for the CAXIF3 element, and four plot elements are generated for the CAXIF4 element. In the last case the elements connect the pairs of points (1-2), (2-3), (3-4), and (4-1).
5. If B = 0.0, the fluid is incompressible.

**CAXISYM**

## Axisymmetric Line Elements Connection - SOL 400

Defines two or three node axisymmetric thick shell elements.

Formats:

| 1       | 2   | 3   | 4  | 5  | 6  | 7    | 8 | 9 | 10 |
|---------|-----|-----|----|----|----|------|---|---|----|
| CAXISYM | EID | PID | G1 | G2 | G3 | NOFF |   |   |    |

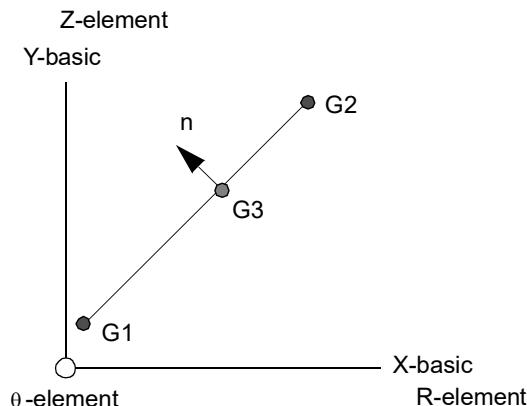
Example:

|         |    |    |   |    |  |  |  |  |  |
|---------|----|----|---|----|--|--|--|--|--|
| CAXISYM | 22 | 98 | 8 | 16 |  |  |  |  |  |
|---------|----|----|---|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of PAXISYM entry. (Integers > 0)  |
| G1, G2     | Identification numbers at the two end grid points. Required data.<br>(Unique; Integers > 0)  |
| G3         | Identification number of the one grid in between G1 and G2. See Remark 2. (Unique<br>Integer $\geq 0$ or blank). If G3=0; this is a straight two node element. |
| NOFF       | Offset from the surface of the grid points to the element reference plane. (Real)  |

Remarks:

1. Element identification numbers should be unique with respect to all other identification numbers.
2. It is recommended that G3 be located within the middle third of the element.
3. The element must lie in the x-y plane of the basic system and is oriented as shown below.



**CBAR****Simple Beam Element Connection**

Defines a simple beam element.

**Format:**

| 1    | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9    | 10 |
|------|-----|-----|-----|-----|-----|-----|-----|------|----|
| CBAR | EID | PID | GA  | GB  | X1  | X2  | X3  | OFFT |    |
|      | PA  | PB  | W1A | W2A | W3A | W1B | W2B | W3B  |    |

**Example:**

|      |   |     |   |   |     |     |     |     |  |
|------|---|-----|---|---|-----|-----|-----|-----|--|
| CBAR | 2 | 39  | 7 | 3 | 0.6 | 18. | 26. | GOG |  |
|      |   | 513 |   |   |     |     |     |     |  |

**Alternate Format and Example:**

|      |     |     |     |     |     |     |     |      |  |
|------|-----|-----|-----|-----|-----|-----|-----|------|--|
| CBAR | EID | PID | GA  | GB  | G0  |     |     | OFFT |  |
|      | PA  | PB  | W1A | W2A | W3A | W1B | W2B | W3B  |  |
| CBAR | 2   | 39  | 7   | 6   | 105 |     |     | GOG  |  |
|      |     | 513 |     |     |     |     |     |      |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PBAR, PBARL or PBRSECT entry.<br>(Integer > 0 or blank*; Default = EID unless BAROR entry has nonzero entry in field 3.)                                      |
| GA, GB     | Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB )   |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 8. (Real)                              |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB) |
| OFFT       | Offset vector interpretation flag. (character or blank) See Remark 8.   |

| Descriptor                     | Meaning   |
|--------------------------------|---|
| PA, PB                         | Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see <a href="#">Figure 9-11</a> ). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.) Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not allowed in SOL 700. |
| W1A, W2A, W3A<br>W1B, W2B, W3B | Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$ , respectively (see <a href="#">Figure 9-11</a> ) in displacement coordinate systems (or in element system depending upon the content of the OFFT field), at points GA and GB, respectively. See Remark 7. and 8. (Real; Default = 0.0) Offsets are not allowed in SOL 700.  |

\*See the BAROR entry for default options for field 3 and fields 6 through 9.

#### Remarks:

- Element identification numbers should be unique with respect to all other element identification numbers.
- [Figure 9-11](#) and [Figure 9-12](#) define bar element geometry with and without offsets:

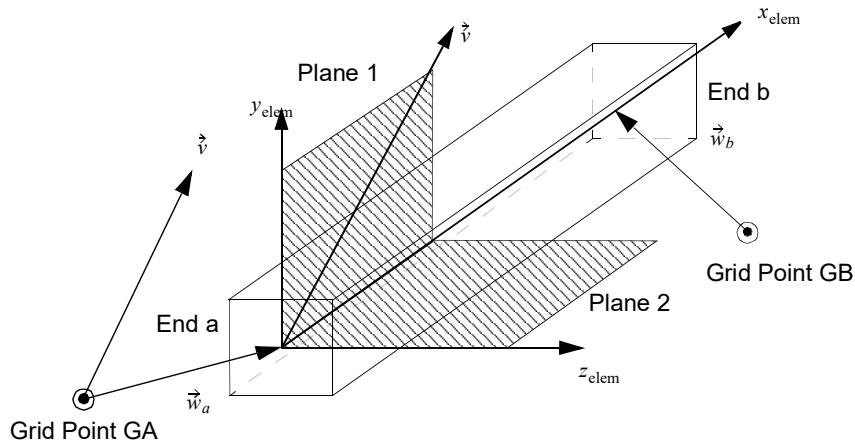


Figure 9-11 CBAR Element Geometry with Offsets

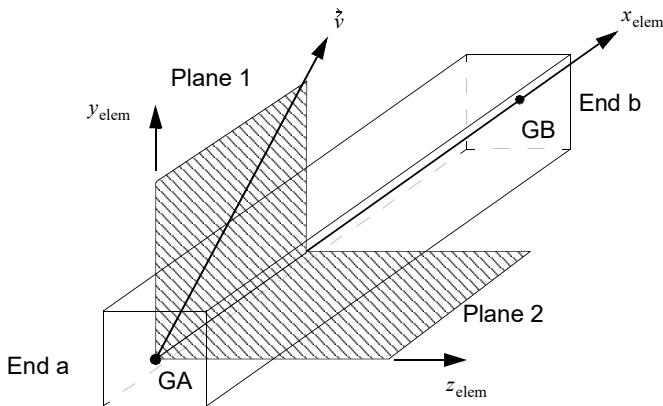


Figure 9-12 CBAR Element Geometry without Offsets

3. [Figure 9-13](#) and [Figure 9-14](#) define the elemental force and moment sign convention.

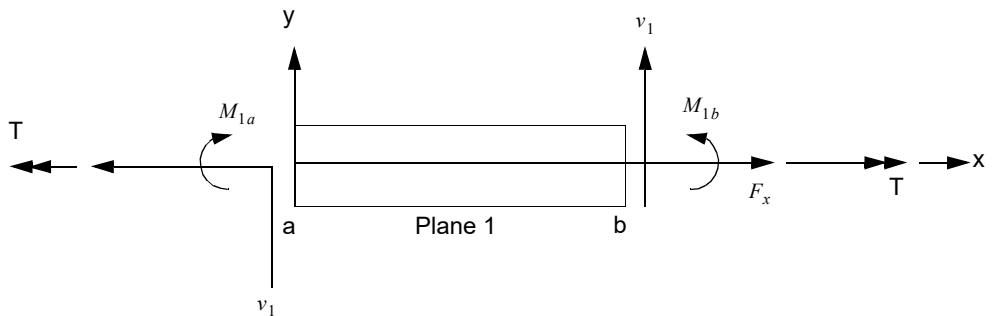


Figure 9-13 CBAR Element Internal Forces and Moments (x-y Plane)

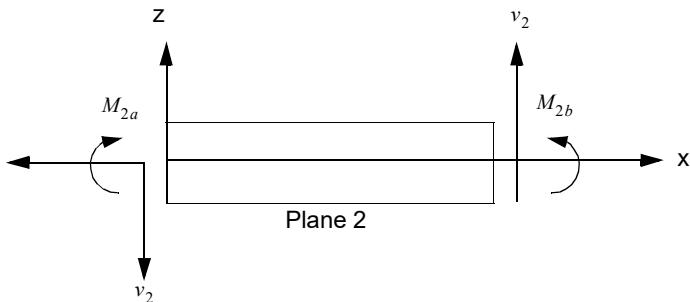


Figure 9-14 CBAR Element Internal Forces and Moments (x-z Plane)

4. The continuation may be omitted if there are no pin flags or offsets.
5. For the case where field 9 is blank and not provided by the BAROR entry, if an integer is specified in field 6, then G0 is used; if field 6 is blank or real, then X1, X2, X3 is used.

6. See [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide* for a definition of coordinate system terminology.
7. Offset vectors are treated internally like rigid elements. For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF.
  - Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.
  - The length of offset vectors is not affected by thermal loads. But the thermal load changes due to location changes by offsets are correctly computed if the enhanced method is used.
  - BAR elements with offsets will give correct buckling results if the enhanced method is used.
  - Masses are correctly transformed for offset effects if MDLPRM, OFFDEF, LROFF is used. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.
  - In nonlinear solution sequences, such as SOL 106 or SOL 400, BAR is treated as linear element. If geometric nonlinear effects are required, please use Bulk Data entry MDLPRM,BRTOBM,1 to convert BAR to BEAM.
8. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

| String | Orientation Vector | End A Offset | End B Offset |
|--------|--------------------|--------------|--------------|
| GGG    | Global             | Global       | Global       |
| BGG    | Basic              | Global       | Global       |
| GGO    | Global             | Global       | Offset       |
| BGO    | Basic              | Global       | Offset       |
| GOG    | Global             | Offset       | Global       |
| BOG    | Basic              | Offset       | Global       |
| GOO    | Global             | Offset       | Offset       |
| BOO    | Basic              | Offset       | Offset       |

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)

9. For SOL 600, the BIT field is ignored unless param,MAROFSET is 1 or 2. An extra flag
10. For SOL 700, the BIT field is ignored and a warning is issued.
11. For RC network solver in thermal analysis, the G0, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B and W3B are ignored.

**CBARAO****Auxiliary Output Points Along Bar Element Axis (CBAR Entry)**

Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output. This entry is applicable in static and normal modes analysis only.

**Format:**

| 1      | 2   | 3     | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-------|----|----|----|----|----|----|----|
| CBARAO | EID | SCALE | X1 | X2 | X3 | X4 | X5 | X6 |    |

**Example:**

|        |      |    |     |     |     |     |  |  |  |
|--------|------|----|-----|-----|-----|-----|--|--|--|
| CBARAO | 1065 | FR | 0.2 | 0.4 | 0.6 | 0.8 |  |  |  |
|--------|------|----|-----|-----|-----|-----|--|--|--|

**Alternate Format and Example:**

|        |      |       |      |     |        |  |  |  |  |
|--------|------|-------|------|-----|--------|--|--|--|--|
| CBARAO | EID  | SCALE | NPTS | X1  | DELTAX |  |  |  |  |
| CBARAO | 1065 | FR    | 4    | 0.2 | 0.2    |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification of a CBAR entry. (0 < Integer < 100,000,000)                     |
| SCALE      | Defines scale of Xi values. (Character = "LE" or "FR")                                  |
| Xi         | Series of locations along element axis for stress and force data recovery. (Real > 0.0) |
| DELTAX     | Incremental distance along element axis. (Real)   |
| NPTS       | Number of stress recovery points, not including the end points. (Integer > 0)           |

**Remarks:**

1. This entry defines intermediate locations on the axis of selected CBAR elements for additional data recovery. The values of Xi are actual distances along the length if SCALE = "LE". If SCALE = "FR", the values of Xi are ratios of actual distances to the bar length. A PLOAD1 Bulk Data entry for the CBAR element in question must be present to obtain intermediate data recovery.
2. When the alternate format is used, a series of locations  $Xi = X[i-1] + DELTAX$ ,  $i = 1, 2, \dots, NPTS$  is generated.
3. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress values at the end points of the beam will always be output. This output format will be used for all beam and bar elements.
4. Intermediate loads on the element defined by the PLOAD1 entry will be accounted for in the calculation of element stresses and forces. If no PLOAD1 entry is defined for the element, the shear forces are constant, the moments are linear, and it is not necessary that the user define additional points.

5. For each bar element, either the basic format or the alternate format, but not both, may be used. A maximum of six internal points can be specified with the basic form. The end points must not be listed because data will be generated for them, as explained in Remark 3. If more than six unequally spaced internal points are desired, it is advisable to subdivide the bar into two or more elements.

**CBEAM****Beam Element Connection**

Defines a beam element.

**Format:**

| 1     | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9        | 10 |
|-------|-----|-----|-----|-----|-----|-----|-----|----------|----|
| CBEAM | EID | PID | GA  | GB  | X1  | X2  | X3  | OFFT/BIT |    |
|       | PA  | PB  | W1A | W2A | W3A | W1B | W2B | W3B      |    |
|       | SA  | SB  |     |     |     |     |     |          |    |

**Example:**

|       |   |     |   |     |     |     |      |     |  |
|-------|---|-----|---|-----|-----|-----|------|-----|--|
| CBEAM | 2 | 39  | 7 | 13  | 8.2 | 6.1 | -5.6 | GOG |  |
|       |   | 513 |   | 3.0 |     |     |      |     |  |
|       | 8 | 5   |   |     |     |     |      |     |  |

**Alternate Format and Example:**

|       |     |     |     |     |     |     |     |          |  |
|-------|-----|-----|-----|-----|-----|-----|-----|----------|--|
| CBEAM | EID | PID | GA  | GB  | G0  |     |     | OFFT/BIT |  |
|       | PA  | PB  | W1A | W2A | W3A | W1B | W2B | W3B      |  |
|       | SA  | SB  |     |     |     |     |     |          |  |
| CBEAM | 2   | 39  | 7   | 13  | 105 |     |     | GOG      |  |
|       |     | 513 |     |     |     |     |     |          |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of PBEAM, PBCOMP, PBEAML or PBMSECT entry. (Integer > 0; Default = EID)*  |
| GA, GB     | Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB )  |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 9. (Real)                                 |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ using grid point G0.<br>Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0;<br>G0 ≠ GA or GB) |
| OFFT       | Offset vector interpretation flag. See Remark 9. (Character or blank)  |
| BIT        | Built-in twist of the cross-sectional axes about the beam axis at end B relative to end A. For beam p-elements only. (Real; Default = 0.0)   |

| Descriptor                     | Meaning   |
|--------------------------------|---|
| PA, PB                         | Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees-of-freedom of the beam. The degrees-of-freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see <a href="#">Figure 9-15</a> ). The beam must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique Integers 1 through 6 with no embedded blanks.) Pin flags are not allowed for beam p-elements. Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not presently allowed in SOL 700. Also, Pin flags should not be used in nonlinear analysis when there is large displacement. |
| W1A, W2A, W3A<br>W1B, W2B, W3B | Components of offset vectors from the grid points to the end points of the axis of the shear center. See Remarks <a href="#">7</a> , <a href="#">8</a> , and <a href="#">9</a> . (Real; Default = 0.0)  |
| SA, SB                         | Scalar or grid point identification numbers for the ends A and B, respectively. The degrees-of-freedom at these points are the warping variables $d\theta/dx$ . SA and SB cannot be specified for beam p-elements. (Integers $\geq 0$ or blank)   |

\*See the BEAMOR entry for default options for field 3 and fields 6 through 9.

### Remarks:

- Element identification numbers should be unique with respect to all other element identification numbers.
- For an additional explanation of the beam element, see the [Beam Element \(CBEAM\)](#) (p. 55) in the *MSC Nastran Reference Guide*. [Figure 9-15](#) defines beam element geometry:

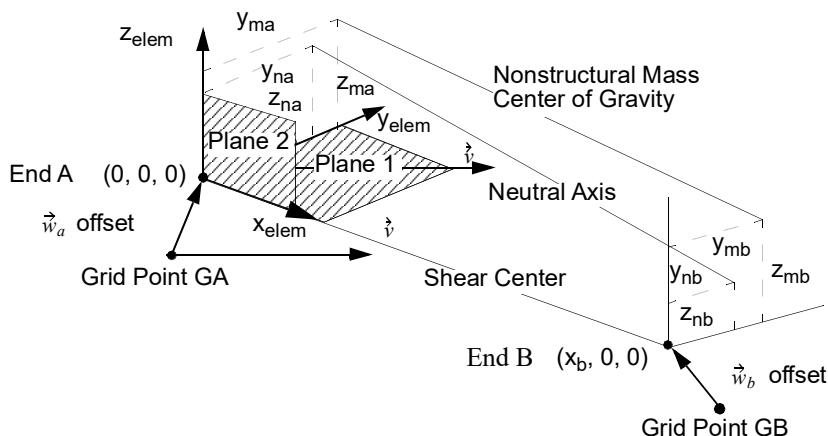


Figure 9-15 CBEAM Element Geometry System (Non p-adaptive)

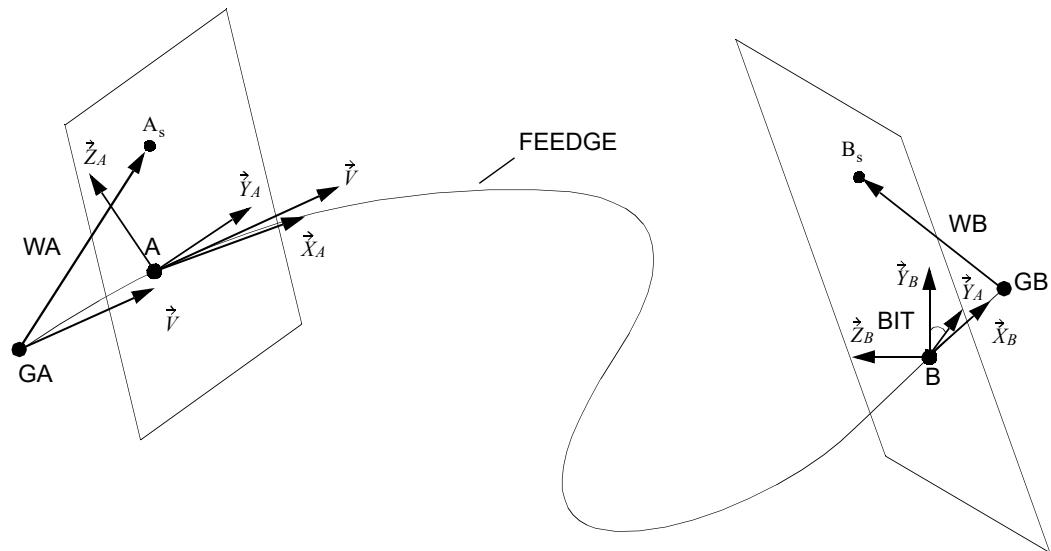


Figure 9-16 CBEAM Element Geometry System (p-adaptive)

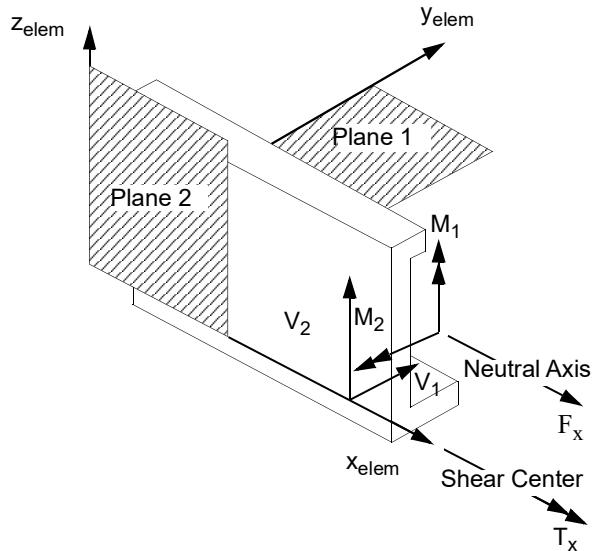


Figure 9-17 CBEAM Internal Element Forces and Moments

3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.
4. G0 cannot be located at GA or GB.
5. The rules for the continuations entries are:
  - Both continuations may be omitted if there are no pin flags, offsets, or warping variables.

- If the second continuation is used, then the first continuation must be included, even if all fields are blank.
  - If the second continuation is omitted, torsional stiffness due to warping of the cross section will not be considered.
6. If warping is allowed ( $SA$  and  $SB > 0$ ), then  $SA$  and  $SB$  must be defined with SPOINT or GRID entries. If GRID entries are used, the warping degree-of-freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for  $SA$  and  $SB$  if GROUNDCHECK is requested in case control.
7. Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.
- For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.
- For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1.
8. If the CBEAM element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
- By default, the edge of the element is considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE entry.
  - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and considered to be straight. Edges with midside nodes cannot be shared by p-elements.
  - For the beam p-element, components of the offset vectors parallel to the beam axis (FEEDGE) will be ignored.
  - For the beam p-element, offset vectors can only be specified in the displacement coordinate systems at the grid points.
9. If the element is a p-version element, BIT in field 9 contains the value of the built-in-twist measured in radians. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

| String | Orientation Vector | End A Offset | End B Offset |
|--------|--------------------|--------------|--------------|
| GGG    | Global             | Global       | Global       |
| BGG    | Basic              | Global       | Global       |
| GGO    | Global             | Global       | Offset       |
| BGO    | Basic              | Global       | Offset       |
| GOG    | Global             | Offset       | Global       |
| BOG    | Basic              | Offset       | Global       |
| GOO    | Global             | Offset       | Offset       |
| BOO    | Basic              | Offset       | Offset       |

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)

10. For RC network solver in thermal analysis, the X1, X2, X3, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B, W3B, SA and SB are ignored.

**CBEAM3****Three-Node Beam Element Connection**

Defines a three-node beam element.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CBEAM3 | EID | PID | GA  | GB  | GC  | X1  | X2  | X3  |    |
|        | W1A | W2A | W3A | W1B | W2B | W3B | W1C | W2C |    |
|        | W3C | TWA | TWB | TWC | SA  | SB  | SC  |     |    |

**Example:**

|        |     |     |     |      |      |      |     |      |  |
|--------|-----|-----|-----|------|------|------|-----|------|--|
| CBEAM3 | 101 | 2   | 201 | 332  | 1000 | 1.0  | 3.5 | -2.0 |  |
|        |     | 3.0 |     | 3.0  | 2.2  | -1.0 |     |      |  |
|        | 2.5 | 10. | 15. | 20.0 | 206  | 301  | 312 |      |  |

**Alternate Format and Example:**

|        |     |     |     |     |     |     |     |     |  |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| CBEAM3 | EID | PID | GA  | GB  | GC  | G0  |     |     |  |
|        | W1A | W2A | W3A | W1B | W2B | W3B | W1C | W2C |  |
|        | W3C | TWA | TWB | TWC | SA  | SB  | SC  |     |  |

|        |     |     |     |      |      |     |     |  |  |
|--------|-----|-----|-----|------|------|-----|-----|--|--|
| CBEAM3 | 101 | 2   | 201 | 332  | 1000 | 105 |     |  |  |
|        |     | 3.0 |     |      | 2.2  | 1.0 |     |  |  |
|        | 2.5 | 10. | 15. | 20.0 | 206  | 301 | 312 |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of PBEAM3 or PBMSECT entries. See Remark 8. (Integer > 0; Required)  |
| GA, GB, GC | Grid point identification numbers of connection points. GA and GB are grid point identification numbers at the two ends of the beam element while GC is the one at the grid point in between. (Integer > 0 or blank; GA, GB and GC must be distinct from each other. See Remark 6.) |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)   |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0; G0 ≠ GA or GB or GC)  |

| Descriptor    | Meaning  |
|---------------|--|
| WiA, WiB, WiC | Components of offsets vectors, measured in the displacement coordinate systems at grid points A, B, and C, from the grid points to the points on the axis of shear center. See Remark 9. (Real; Default = 0.0) |
| TWA, TWB, TWC | Pretwist angles in degrees at A, B, and C, respectively. (Real; Default = 0.0)   |
| SA, SB, SC    | Scalar or grid point identification numbers for A, B, and C, respectively. The degrees of freedom at these points are warping variables. (Integer > 0 or blank)  |

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If field 7 is an integer, then G0 is used. If field 7 is blank or real, then X1, X2, X3 are used.
3. G0 cannot be located at GA or GB or GC.
4. If warping effect is included in the analysis (SA, SB and SC > 0), then SA, SB, and SC must be defined with either SPOINT or GRID entries. If GRID entries are used, the warping degree of freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for SA, SB and SC if GROUNDCHECK is requested in case control.
5. BEAMOR cannot be used to set up default options for field 3 and fields 6 through 8 for CBEAM3 entries.
6. If GC is left blank, then the element degenerates to a formulation similar to the two-node straight beam element. The two-node straight beam formulation is not the CBEAM formulation and will result in overly stiff results with warping and shear center offset. This is because the CBEAM3 is a Variational Asymptotic Beam Section that generalizes Timoshenko beam theory and requires the mid node for accuracy.
7. This entry is not available in SOL 600.
8. For CBEAM3 referencing PBMSECT, the CORE/LAYER keywords are required or a Fatal message is issued. For this composite case, it is recommended that CBEAM3 has 3 nodes and 3 warping DOFs.

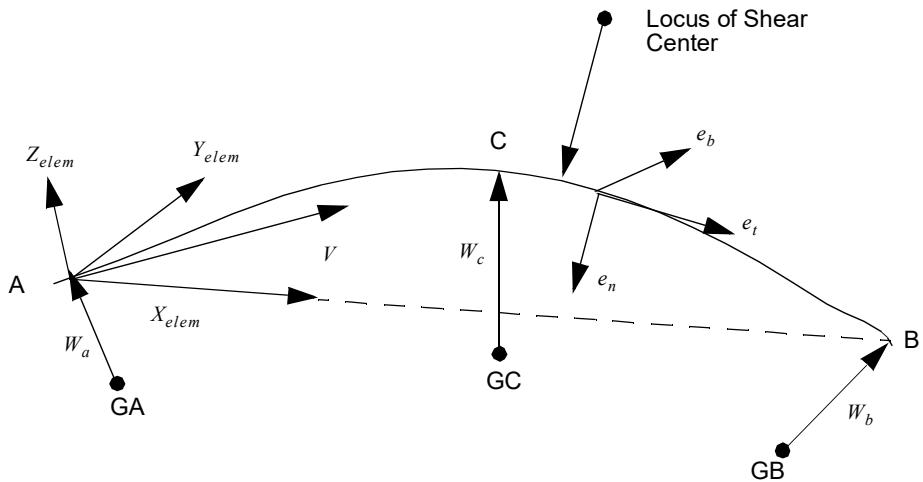


Figure 9-18 CBEAM3 Element Geometry System

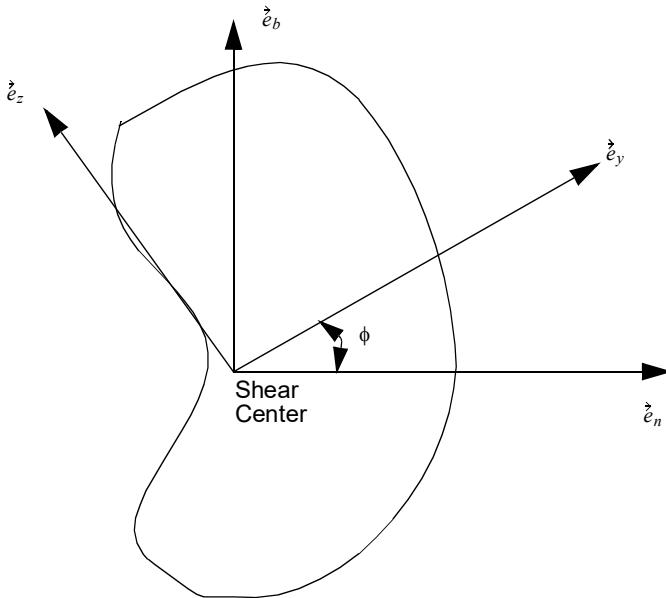


Figure 9-19 Local Coordinate System on Beam Cross-Section

- Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

**CBEND****Curved Beam or Pipe Element Connection**

Defines a curved beam, curved pipe, or elbow element.

**Format:**

| 1     | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9    | 10 |
|-------|-----|-----|----|----|----|----|----|------|----|
| CBEND | EID | PID | GA | GB | X1 | X2 | X3 | GEOM |    |

**Example:**

|       |    |    |    |    |     |     |      |   |  |
|-------|----|----|----|----|-----|-----|------|---|--|
| CBEND | 32 | 39 | 17 | 19 | 6.2 | 5.1 | -1.2 | 3 |  |
|-------|----|----|----|----|-----|-----|------|---|--|

**Alternate Format and Example:**

|       |     |     |    |    |     |  |  |      |  |
|-------|-----|-----|----|----|-----|--|--|------|--|
| CBEND | EID | PID | GA | GB | G0  |  |  | GEOM |  |
| CBEND | 32  | 39  | 17 | 19 | 106 |  |  | 3    |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PBEND entry. (Integer > 0; Default = EID)   |
| GA, GB     | Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB )   |
| X1, X2, X3 | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)   |
| G0         | Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB) |
| GEOM       | Flag to select specification of the bend element. See Remark 3. (1 ≤ Integer ≤ 4; No Default)   |

**Remarks:**

- Element identification numbers must be unique with respect to all other element identification numbers.

2. For an additional explanation of the CBEND element, see the PBEND entry description. [Figure 9-20](#) and [Figure 9-21](#) define the element coordinate system and internal forces and moments.

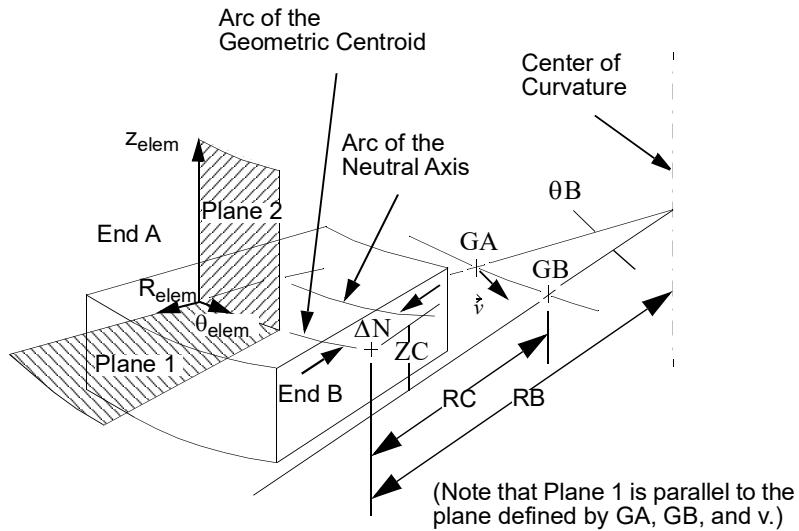


Figure 9-20 CBEND Element Coordinate System

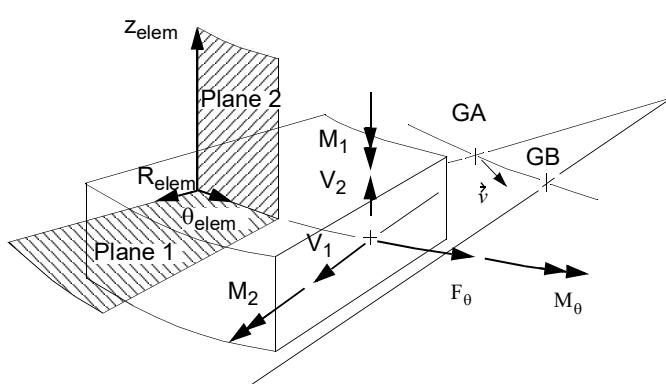
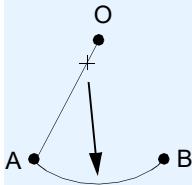
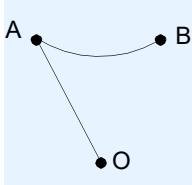
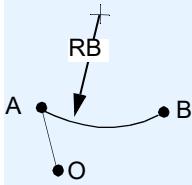
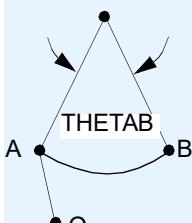


Figure 9-21 CBEND Element Internal Forces and Moments

3. The options for element connection to GA, GB using GEOM are the following.

Table 2      GEOM Options

| Configuration  | GEOM | Description  |
|--|------|--|
|   | 1    | The center of curvature lies on the line AO (or its extension) or vector $\vec{v}$ .   |
|   | 2    | The tangent of centroid arc at end A is parallel to line AO or vector $\vec{v}$ . Point O (or vector $\vec{v}$ ) and the arc $\overline{AB}$ must be on the same side of the chord $\overline{AB}$ .   |
|   | 3    | The bend radius (RB) is specified on the PBEND entry: Points A, B, and O (or vector $\vec{v}$ ) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$ ) lies on the opposite side of line AB from the center of curvature. |
|  | 4    | THETAB is specified on the PBEND entry. Points A, B, and O (or vector $\vec{v}$ ) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$ ) lies on the opposite side of line AB from the center of curvature.               |

4. For RC network solver in thermal analysis, the X1, X2, X3 and GEOM are ignored.

**CBUSH**

## Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

**Format:**

| 1     | 2   | 3    | 4  | 5  | 6     | 7  | 8  | 9   | 10 |
|-------|-----|------|----|----|-------|----|----|-----|----|
| CBUSH | EID | PID  | GA | GB | GO/X1 | X2 | X3 | CID |    |
|       | S   | OCID | S1 | S2 | S3    |    |    |     |    |

**Example 1:** Noncoincident grid points.

|       |    |   |   |     |    |  |  |  |  |
|-------|----|---|---|-----|----|--|--|--|--|
| CBUSH | 39 | 6 | 1 | 100 | 75 |  |  |  |  |
|-------|----|---|---|-----|----|--|--|--|--|

**Example 2:** GB not specified.

|       |    |   |   |  |  |  |  |   |  |
|-------|----|---|---|--|--|--|--|---|--|
| CBUSH | 39 | 6 | 1 |  |  |  |  | 0 |  |
|-------|----|---|---|--|--|--|--|---|--|

**Example 3:** Coincident grid points (GA ≠ GB).

|       |    |   |   |  |  |  |  |   |  |
|-------|----|---|---|--|--|--|--|---|--|
| CBUSH | 39 | 6 | 1 |  |  |  |  | 6 |  |
|-------|----|---|---|--|--|--|--|---|--|

**Example 4:** Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

|       |      |    |    |     |     |  |  |  |  |
|-------|------|----|----|-----|-----|--|--|--|--|
| CBUSH | 39   | 6  | 1  | 600 |     |  |  |  |  |
|       | 0.25 | 10 | 0. | 10. | 10. |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )  |
| PID        | Property identification number of a PBUSH entry. ( $\text{Integer} > 0$ ; Default = EID)   |
| GA, GB     | Grid point identification number of connection points. See Remark 6. ( $GA > 0$ , $GB \geq 0$ or blank)  |
| Xi         | Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)  |
| GO         | Alternate method to supply vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to End A. See Remark 3. (Integer $> 0$ )   |
| CID        | Element coordinate system identification. A 0 value means the basic coordinate system will be used. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 9-22 and Remark 3. (Integer $\geq 0$ or blank) |
| S          | Location of spring damper. See Figure 9-21. (Real; Default = 0.5)  |

| Descriptor | Meaning   |
|------------|---|
| OCID       | Coordinate system identification of spring-damper offset. See Remark 9. (Integer $\geq -1$ ; Default = -1, which means the offset point lies on the line between GA and GB according to <a href="#">Figure 9-22</a> ) |
| S1, S2, S3 | Components of spring-damper offset in the OCID coordinate system if OCID $\geq 0$ . See <a href="#">Figure 9-23</a> and Remark 9. (Real)  |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. [Figure 9-22](#) shows the bush element geometry.
3. CID  $\geq 0$  overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis. Nastran does not convect user specified coordinate systems, thus with this option, in nonlinear analysis the element stiffness direction does not change with deformation.
4. For noncoincident grids ( $GA \neq GB$ ), when GO or ( $X1, X2, X3$ ) is given and no CID is specified, the line AB is the element x-axis and the orientation vector  $\vec{v}$  lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids ( $GA \neq GB$ ), if neither GO or ( $X1, X2, X3$ ) is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is a grounded terminal, a grounded terminal is a point with a displacement that is constrained to zero.
7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.
8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID  $\geq 0$ , then S is ignored and S1, S2, S3 are used.

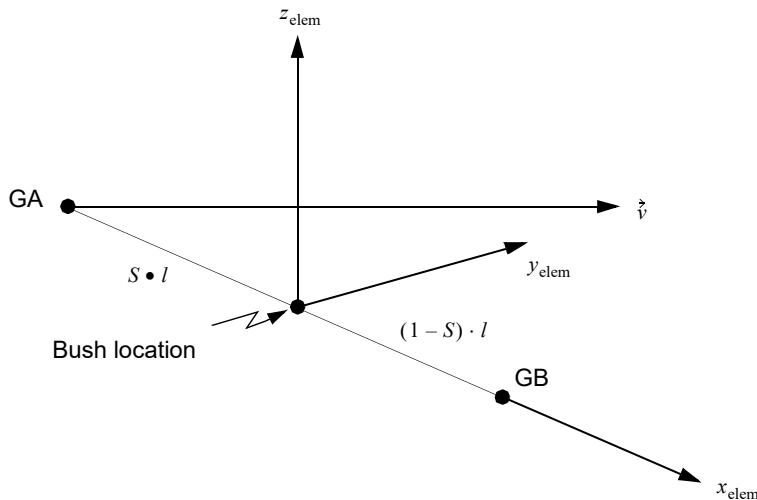
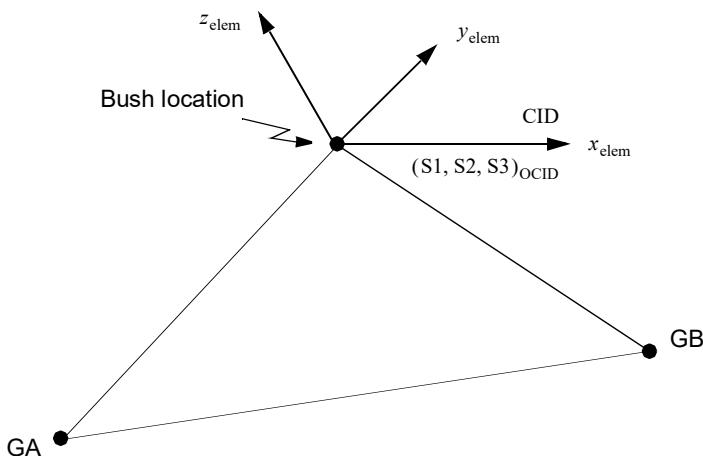


Figure 9-22 CBUSH Element



The material stiffness and damping properties of the elastomer are located at  $(S_1, S_2, S_3)$ .

Figure 9-23 Definition of Offset S1, S2, S3

10. When  $\text{CID} \geq 0$ , the element x-axis is set as in Remark 3. This means that the element force is always computed as  $K_e \cdot (U_B - U_A)$ ; if  $U_A > U_B$ , a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.
11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.

12. For SOL 600, if G0, X1, X2, X3, CID or OCID are entered, a Severe Warning will be issued and Marc will not run. SOL 600 translates the spring and damping terms in global coordinates of GA and GB and will use K1 to K6 and B1 to B6 whether or not GA and GB are coincident or not. S, S1, S2, S3 are ignored. CID is ignored unless it is zero, in which case K1 and B1 are along the axis of GA to GB and K2-K6 and B2-B6 are ignored if entered.
13. CBUSH elements are not supported in thermal analysis.

**CBUSH1D**

## Rod Type Spring-and-Damper Connection

Defines the connectivity of a one-dimensional spring and viscous damper element.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6   | 7 | 8 | 9 | 10 |
|---------|-----|-----|----|----|-----|---|---|---|----|
| CBUSH1D | EID | PID | GA | GB | CID |   |   |   |    |

**Example:**

|         |    |     |     |     |  |  |  |  |  |
|---------|----|-----|-----|-----|--|--|--|--|--|
| CBUSH1D | 35 | 102 | 108 | 112 |  |  |  |  |  |
|---------|----|-----|-----|-----|--|--|--|--|--|

| Descriptor | Meaning   | Default Values |
|------------|---|----------------|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )       | Required       |
| PID        | Property identification number of a PBUSH1D entry. ( $\text{Integer} > 0$ ) | EID            |
| GA         | Grid point id of first grid.  | Required       |
| GB         | Grid point id of second grid  | blank          |
| CID        | Coordinate system id. ( $\text{Integer} \geq 0$ )                           | blank          |

**Remarks:**

1. For noncoincident grids  $GA \neq GB$  and if CID is blank, the line GA to GB is the element axis. In geometric nonlinear analysis, the element axis (line GA to GB) follows the deformation of grids GA and GB. See [Figure 9-24](#).
2. If  $CID \geq 0$  is specified, the x-axis of the CID coordinate system is the element axis. In geometric nonlinear analysis, the element axis (x-axis of CID) remains fixed.
3. If GA and GB are coincident or if GB is blank, then  $CID \geq 0$  must be specified and the element axis is the x-axis of CID.

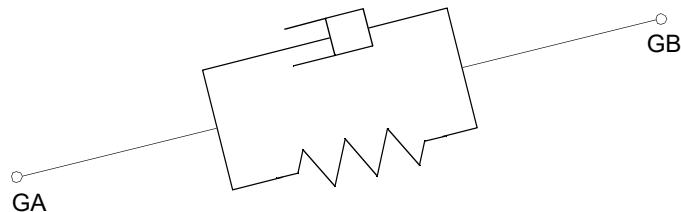


Figure 9-24 Spring and Damper Element

**CBUSH2D****2-D Linear-Nonlinear Connection**

Defines the connectivity of a two-dimensional Linear-Nonlinear element.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6   | 7     | 8 | 9 | 10 |
|---------|-----|-----|----|----|-----|-------|---|---|----|
| CBUSH2D | EID | PID | GA | GB | CID | PLANE |   |   |    |

**Example:**

|         |     |     |      |      |   |    |  |  |  |
|---------|-----|-----|------|------|---|----|--|--|--|
| CBUSH2D | 100 | 101 | 1001 | 2001 | 0 | XY |  |  |  |
|---------|-----|-----|------|------|---|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ ; Required)      |
| PID        | Property identification number of a PBUSH2D. ( $\text{Integer} > 0$ ; Required)      |
| GA         | Inner grid. ( $\text{Integer} > 0$ ; Required)                                       |
| GB         | Outer grid. ( $\text{Integer} > 0$ ; Required)                                       |
| CID        | Coordinate system used to define 2-D plane. ( $\text{Integer} \geq 0$ ; Default = 0) |
| PLANE      | Orientation plane in CID: XY, YZ, ZX, see Remark 1. (Character; Default = 'XY')      |

**Remarks:**

1. The XY, YZ, and ZX planes are relative to the displacements coordinates of GA and GB. The planes correspond to directions 1 and 2. GA and GB should be coincident grids with parallel displacement coordinate systems. The coordinate systems are not checked. Wrong answers will be produced if this rule is not followed.
2. The behavior is different if there is no rotor in the model and a CBUSH2D is used. When this happens, for most cases, if there is no rotor the nominal stiffness values from the PBUSH2D will be used, with no frequency-dependence. However, if an ELEMUDS is used, the CBUSH2D will be ignored if there is no rotor in the model.

**CCONEAX**

## Axisymmetric Shell Element Connection

Defines a conical shell element.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-----|----|----|---|---|---|---|----|
| CCONEAX | EID | PID | RA | RB |   |   |   |   |    |

**Example:**

|         |   |   |   |   |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|
| CCONEAX | 1 | 2 | 3 | 4 |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)               |
| PID        | Property identification number of a PCONEAX entry. (Integer > 0; Default = EID) |
| RA         | Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB )                |
| RB         | Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB )                |

**Remarks:**

1. This element has limited capabilities. See the *MSC Nastran Reference Guide*, Section 5.3.3.
2. This entry is allowed only if an AXIC entry is also present.
3. In order to reference this entry on a SET Case Control command, the ID must be modified by

$$\text{ID}_n = \text{ID} \cdot 1000 + n$$

where n is the harmonic number plus one and ID<sub>n</sub> is the value specified on the SET entry.

**CDAMP1**

## Scalar Damper Connection

Defines a scalar damper element.

Format:

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|----|----|----|----|---|---|----|
| CDAMP1 | EID | PID | G1 | C1 | G2 | C2 |   |   |    |

Example:

|        |    |   |   |  |    |   |  |  |  |
|--------|----|---|---|--|----|---|--|--|--|
| CDAMP1 | 19 | 6 | 0 |  | 23 | 2 |  |  |  |
|--------|----|---|---|--|----|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )                      |
| PID        | Property identification number of a PDAMP property entry. ( $\text{Integer} > 0$ ; Default = EID) |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )                           |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point.)              |

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

**CDAMP1D**

## Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

**Format:**

| 1       | 2    | 3      | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|------|--------|----|----|----|----|---|---|----|
| CDAMP1D | EID  | PID    | G1 | C1 | G2 | C2 |   |   |    |
|         | CORD | FOLLOW |    |    |    |    |   |   |    |

**Example:**

|         |      |     |    |   |  |  |  |  |  |
|---------|------|-----|----|---|--|--|--|--|--|
| CDAMP1D | 1001 | 101 | 55 | 1 |  |  |  |  |  |
|---------|------|-----|----|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )  |
| PID        | Property identification number of a PDAMPn entry. ( $\text{Integer} > 0$ ; Default = EID)   |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )   |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.) |
| CORD       | Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ( $\text{Integer} \geq 0$ )  |
| FOLLOW     | Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.  |

**Remark:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

**CDAMP2**

## Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|---|----|----|----|----|---|---|----|
| CDAMP2 | EID | B | G1 | C1 | G2 | C2 |   |   |    |

**Example:**

|        |    |      |    |   |  |  |  |  |  |
|--------|----|------|----|---|--|--|--|--|--|
| CDAMP2 | 16 | 2.98 | 32 | 1 |  |  |  |  |  |
|--------|----|------|----|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )         |
| B          | Value of the scalar damper. (Real)   |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )              |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point.) |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
8. RC network solver does not support CDAMP2 for thermal analysis.

**CDAMP2D**

Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

**Format:**

| 1       | 2    | 3      | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|------|--------|----|----|----|----|---|---|----|
| CDAMP2D | EID  | B      | G1 | C1 | G2 | C2 |   |   |    |
|         | CORD | FOLLOW |    |    |    |    |   |   |    |

**Example:**

|         |      |     |    |   |  |  |  |  |  |
|---------|------|-----|----|---|--|--|--|--|--|
| CDAMP2D | 1001 | 101 | 55 | 1 |  |  |  |  |  |
|---------|------|-----|----|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )  |
| B          | Value of the scalar damper. (Real)  |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )   |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.) |
| CORD       | Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ( $\text{Integer} \geq 0$ )  |
| FOLLOW     | Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.  |

**Remark:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

**CDAMP3**

## Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|----|----|---|---|---|---|----|
| CDAMP3 | EID | PID | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |     |    |    |  |  |  |  |  |
|--------|----|-----|----|----|--|--|--|--|--|
| CDAMP3 | 16 | 978 | 24 | 36 |  |  |  |  |  |
|--------|----|-----|----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)             |
| PID        | Property identification number of a PDAMP entry. (Integer > 0; Default = EID) |
| S1, S2     | Scalar point identification numbers. (Integer $\geq 0$ ; S1 $\neq$ S2 )       |

**Remarks:**

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP3 for thermal analysis.

**CDAMP4****Scalar Damper Property and Connection to Scalar Points Only**

Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|----|----|---|---|---|---|----|
| CDAMP4 | EID | B | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |      |   |   |  |  |  |  |  |
|--------|----|------|---|---|--|--|--|--|--|
| CDAMP4 | 16 | -2.6 | 4 | 9 |  |  |  |  |  |
|--------|----|------|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )    |
| B          | Scalar damper value. (Real)   |
| S1, S2     | Scalar point identification numbers. ( $\text{Integer} \geq 0$ ; $S1 \neq S2$ ) |

**Remarks:**

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP4 for thermal analysis.

**CDAMP5****Scalar Damper with Material Property**

Defines a damping element that refers to a material property entry and connection to grid or scalar points. This element is intended for heat transfer analysis only.

Format:

| 1      | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|----|----|---|---|---|---|----|
| CDAMP5 | EID | PID | G1 | G2 |   |   |   |   |    |

Example:

|        |   |   |    |    |  |  |  |  |  |
|--------|---|---|----|----|--|--|--|--|--|
| CDAMP5 | 1 | 4 | 10 | 20 |  |  |  |  |  |
|--------|---|---|----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )                            |
| PID        | Identification number of a PDAMP5 property entry. ( $\text{Integer} > 0$ ; Default = EID)               |
| G1, G2     | Grid or scalar point identification numbers. ( $\text{Integer} \geq 0$ and $\text{G1} \neq \text{G2}$ ) |

Remarks:

1. G1 or G2 may be blank or zero indicating a constraint.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. CDAMP5 generates a lumped heat capacity in heat transfer analysis.
4. A scalar point specified on CDAMP5 need not be defined on an SPOINT entry.
5. This entry is not supported in SOL 600.
6. RC network solver does not support CDAMP5 for thermal analysis.

## CDUMi

## Dummy Element Connection

Defines a dummy element ( $3 \leq i \leq 7$ ).

Format:

| 1     | 2   | 3   | 4      | 5  | 6  | 7  | 8      | 9 | 10 |
|-------|-----|-----|--------|----|----|----|--------|---|----|
| CDUMi | EID | PID | G1     | G2 | G3 | G4 | -etc.- |   |    |
|       | A1  | A2  | -etc.- |    |    |    |        |   |    |

Example:

|       |     |     |      |   |   |    |    |  |  |
|-------|-----|-----|------|---|---|----|----|--|--|
| CDUM3 | 114 | 108 | 2    | 5 | 6 | 8  | 11 |  |  |
|       | 2.4 |     | 3.E4 | 2 |   | 50 |    |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)                                  |
| PID        | Property identification number of a PDUMi entry. See Remark 2. (Integer > 0)                |
| Gi         | Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2<br>... ≠ GN ) |
| Ai         | Additional fields. (Real or Integer)  |

Remarks:

1. The user must write the associated element subroutines for matrix generation, stress recovery, etc., and perform a link edit to replace the dummy routines. See the *MSC Nastran Programmer's Manual*.
2. If no property entry is required, PID may contain the material identification number.
3. Additional entries are defined in the user-written element routines.
4. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran". Other than the EID field, all field checking is the responsibility of the user supplied code.

**CELAS1**

## Scalar Spring Connection

Defines a scalar spring element.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|----|----|----|----|---|---|----|
| CELAS1 | EID | PID | G1 | C1 | G2 | C2 |   |   |    |

**Example:**

|        |   |   |  |  |   |   |  |  |  |
|--------|---|---|--|--|---|---|--|--|--|
| CELAS1 | 2 | 6 |  |  | 8 | 1 |  |  |  |
|--------|---|---|--|--|---|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )             |
| PID        | Property identification number of a PELAS entry. ( $\text{Integer} > 0$ ; Default = EID) |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )                  |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point.)     |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. It is recommended that for most applications the grids connected by CELAS1 entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

**CELAS1D**

Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

**Format:**

| 1       | 2    | 3      | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|------|--------|----|----|----|----|---|---|----|
| CELAS1D | EID  | PID    | G1 | C1 | G2 | C2 |   |   |    |
|         | CORD | FOLLOW |    |    |    |    |   |   |    |

**Example:**

|         |      |     |    |   |   |   |  |  |  |
|---------|------|-----|----|---|---|---|--|--|--|
| CELAS1D | 1001 | 101 | 55 | 1 | 8 | 1 |  |  |  |
|---------|------|-----|----|---|---|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )   |
| PID        | Property identification number of a PELAS entry. ( $\text{Integer} > 0$ ; Default = EID)   |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )  |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.) |
| CORD       | Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ( $\text{Integer} \geq 0$ ) (SOL 700 only)  |
| FOLLOW     | Method to update the direction vector in which the spring acts:<br>For SOL 700:<br>FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.                              |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Available in SOL 700 only.
8. It is recommended that for most applications the grids connected by CELAS1D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

**CELAS2****Scalar Spring Property and Connection**

Defines a scalar spring element without reference to a property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|-----|---|----|----|----|----|----|---|----|
| CELAS2 | EID | K | G1 | C1 | G2 | C2 | GE | S |    |

**Example:**

|        |    |       |    |  |    |   |  |  |  |
|--------|----|-------|----|--|----|---|--|--|--|
| CELAS2 | 28 | 6.2+3 | 32 |  | 19 | 4 |  |  |  |
|--------|----|-------|----|--|----|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )           |
| K          | Stiffness of the scalar spring. (Real)   |
| G1, G2     | Geometric grid point or scalar identification number. ( $\text{Integer} \geq 0$ )      |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 blank or zero if scalar point.) |
| GE         | Damping coefficient. See Remarks 6. and 8. (Real)                                      |
| S          | Stress coefficient. (Real)   |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.
8. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.

**CELAS2D****Scalar Spring Connection for SOL 700 Only**

Defines a scalar spring connection for use in SOL 700 only.

**Format:**

| 1       | 2    | 3      | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|---------|------|--------|----|----|----|----|---|---|----|
| CELAS2D | EID  | K      | G1 | C1 | G2 | C2 |   |   |    |
|         | CORD | FOLLOW |    |    |    |    |   |   |    |

**Example:**

|         |      |     |    |   |   |   |  |  |  |
|---------|------|-----|----|---|---|---|--|--|--|
| CELASD2 | 1001 | 101 | 55 | 1 | 8 | 1 |  |  |  |
|---------|------|-----|----|---|---|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )   |
| K          | Stiffness of the scalar spring. (Real)   |
| G1, G2     | Geometric grid point identification number. ( $\text{Integer} \geq 0$ )  |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.) |
| CORD       | Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ( $\text{Integer} \geq 0$ )   |
| FOLLOW     | Method to update the direction vector in which the spring acts:<br>FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.  |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

7. It is recommended that for most applications the grids connected by CELAS2D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

**CELAS3****Scalar Spring Connection to Scalar Points Only**

Defines a scalar spring element that connects only to scalar points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|----|----|---|---|---|---|----|
| CELAS3 | EID | PID | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |   |    |    |  |  |  |  |  |
|--------|----|---|----|----|--|--|--|--|--|
| CELAS3 | 19 | 2 | 14 | 15 |  |  |  |  |  |
|--------|----|---|----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)             |
| PID        | Property identification number of a PELAS entry. (Integer > 0; Default = EID) |
| S1, S2     | Scalar point identification numbers. (Integer $\geq 0$ ; S1 $\neq$ S2 )       |

**Remarks:**

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CELAS4**

## Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|----|----|---|---|---|---|----|
| CELAS4 | EID | K | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |       |   |  |  |  |  |  |  |
|--------|----|-------|---|--|--|--|--|--|--|
| CELAS4 | 42 | 6.2-3 | 2 |  |  |  |  |  |  |
|--------|----|-------|---|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)       |
| K          | Stiffness of the scalar spring. (Real)                                  |
| S1, S2     | Scalar point identification numbers. (Integer $\geq 0$ ; S1 $\neq$ S2 ) |

**Remarks:**

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. A structural damping coefficient is not available with CELAS4. The value of g is assumed to be 0.0.
4. No stress coefficient is available with CELAS4.
5. Only one scalar spring element may be defined on a single entry.
6. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
7. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CFAST****A Shell Patch Fastener Connection**

Defines a fastener with material orientation connecting two surface patches. Large displacement and large rotational effects are supported when in SOL 600 and SOL 400.

Format:

| 1     | 2   | 3   | 4    | 5   | 6   | 7  | 8  | 9  | 10 |
|-------|-----|-----|------|-----|-----|----|----|----|----|
| CFAST | EID | PID | TYPE | IDA | IDB | GS | GA | GB |    |
|       | XS  | YS  | ZS   |     |     |    |    |    |    |

Example using PROP:

|       |   |    |      |    |    |     |  |  |  |
|-------|---|----|------|----|----|-----|--|--|--|
| CFAST | 3 | 20 | PROP | 21 | 24 | 206 |  |  |  |
|-------|---|----|------|----|----|-----|--|--|--|

Example using ELEM:

|       |   |    |      |    |    |     |  |  |  |
|-------|---|----|------|----|----|-----|--|--|--|
| CFAST | 7 | 70 | ELEM | 27 | 74 | 707 |  |  |  |
|-------|---|----|------|----|----|-----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PFAST entry. (Integer > 0; Default = EID)  |
| TYPE       | Specifies the surface patch definition: (Character)<br>If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. See Remark 1. and <a href="#">Figure 9-25</a> .<br>If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. See Remark 1. and <a href="#">Figure 9-25</a> . |
| IDA, IDB   | Property id (for PROP option) or Element id (for ELEM option) defining patches A and B. IDA ≠ IDB (Integer > 0)  |
| GS         | Grid point defining the location of the fastener. See Remark 2. (Integer > 0 or blank)   |
| GA, GB     | Grid ids of piercing points on patches A and B. See Remark 2. (Integer > 0 or blank)   |
| XS, YS, ZS | Location of the fastener in basic. Required if neither GS nor GA is defined. See Remark 2. (Real or blank)   |

Remarks:

1. The CFAST defines a flexible connection between two surface patches. The number of unique physical grids connected depends on the location for the piercing points GA and GB and the size of the diameter D (see [PFAST, 2829](#)).

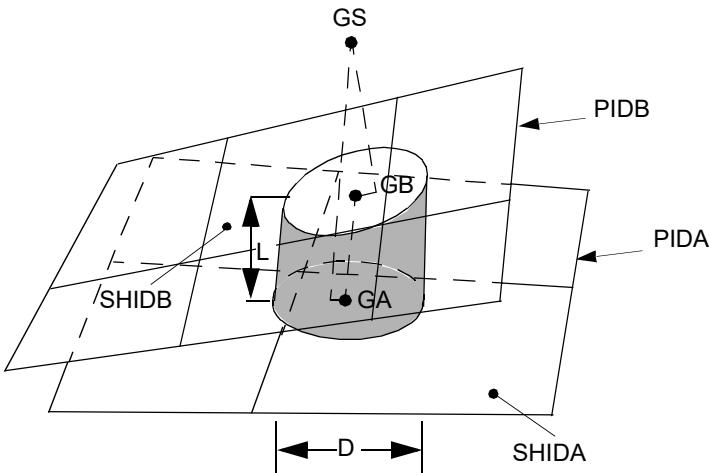


Figure 9-25 Patches Defined with TYPE= 'PROP' or TYPE= 'ELEM'

2. GS defines the approximate location of the fastener in space. GS is projected onto the surface patches A and B. The resulting piercing points GA and GB define the axis of the fastener. GS does not have to lie on the surfaces of the patches. GS must be able to project normals to the two patches. GA can be specified in lieu of GS, in which case GS will be ignored. If neither GS nor GA is specified, then (XS, YS, ZS) in basic must be specified.

If both GA and GB are specified, they must lie on or at least have projections onto surface patches A and B respectively. If GA and GB are both specified, GS is ignored. By default, the locations of user specified GA and GB will not be changed. If the user specifies "SWLDP RM, MOVGAB, 1,", then the locations will be corrected so that they lie on the surface patches A and B within machine precision. The length of the fastener is the final distance between GA and GB. If the length is zero, the normal to patch A is used to define the axis of the fastener.

Diagnostic printouts, checkout runs and control of search and projection parameters are requested on the SWLDP RM Bulk Data entry.

3. The use of param,cfdiagp,yes and param,cfranel,real\_fraction\_value allows for the random removal of a percentage of CFAST elements for failure studies.
4. This entry is not supported in SOL 700.
5. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
6. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
7. If partitioned superelements are present, then CFAST is supported in the main Bulk Data section only.

**CFLUIDi****Fluid Element Connections**

Defines three types of fluid elements for an axisymmetric fluid model.

**Formats:**

| 1       | 2   | 3    | 4    | 5    | 6    | 7   | 8 | 9 | 10 |
|---------|-----|------|------|------|------|-----|---|---|----|
| CFLUID2 | EID | IDF1 | IDF2 |      |      | RHO | B |   |    |
| CFLUID3 | EID | IDF1 | IDF2 | IDF3 |      | RHO | B |   |    |
| CFLUID4 | EID | IDF1 | IDF2 | IDF3 | IDF4 | RHO | B |   |    |

**Examples:**

|         |     |    |    |    |    |      |     |  |  |
|---------|-----|----|----|----|----|------|-----|--|--|
| CFLUID2 | 100 | 11 | 14 |    |    | .025 | 0.0 |  |  |
| CFLUID3 | 110 | 15 | 13 | 12 |    | 1.2  |     |  |  |
| CFLUID4 | 120 | 11 | 15 | 12 | 14 |      |     |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)                                     |
| IDFi       | Identification number of a RINGFL entry. (Integer > 0; IDF1 ≠ IDF2 ≠ IDF3 ≠ IDF4 ; all IDFi < 500000) |
| RHO        | Mass density. (Real > 0.0; Default is the value of DRHO on the AXIF entry)                            |
| B          | Bulk modulus, pressure per volume ratio. (Real; Default is the value of DB on the AXIF entry)         |

**Remarks:**

1. CFLUIDi is allowed only if an AXIF entry is also present.
2. Element identification number must be unique with respect to all other fluid, scalar, and structural elements.

3. The volume defined by IDF<sub>i</sub> is a body of revolution about the polar axis of the fluid coordinate system defined by AXIF. CFLUID2 defines a thick disk with IDF1 and IDF2 defining the outer corners as shown in [Figure 9-26](#):

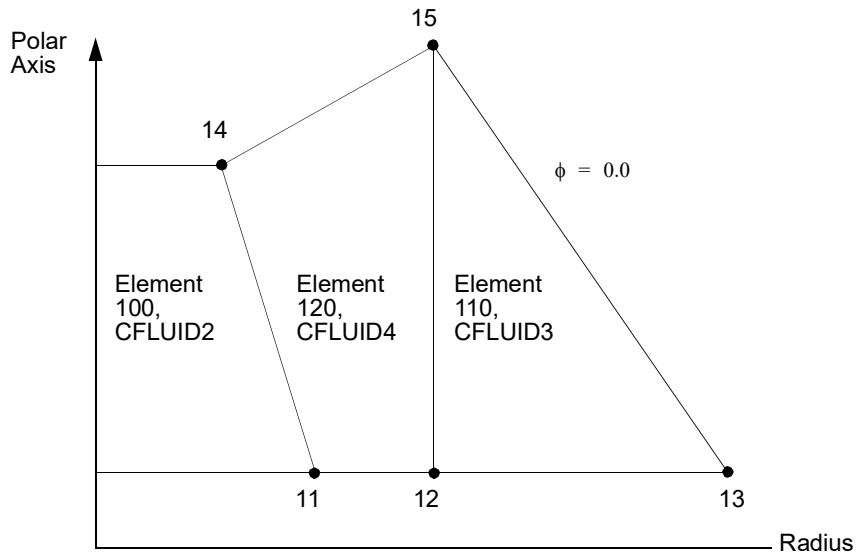


Figure 9-26 CFLUIDi Examples

4. All interior angles must be less than 180°.
5. The order of connected RINGFL points is arbitrary.
6. If B = 0.0, the fluid is incompressible.

**CGAP****Gap Element Connection**

Defines a gap or friction element.

**Format:**

| 1    | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9   | 10 |
|------|-----|-----|----|----|----|----|----|-----|----|
| CGAP | EID | PID | GA | GB | X1 | X2 | X3 | CID |    |

**Example:**

|      |    |   |     |     |     |     |      |  |  |
|------|----|---|-----|-----|-----|-----|------|--|--|
| CGAP | 17 | 2 | 110 | 112 | 5.2 | 0.3 | -6.1 |  |  |
|------|----|---|-----|-----|-----|-----|------|--|--|

**Alternate Format and Example:**

|      |     |     |     |     |    |  |  |     |  |
|------|-----|-----|-----|-----|----|--|--|-----|--|
| CGAP | EID | PID | GA  | GB  | GO |  |  | CID |  |
| CGAP | 17  | 2   | 110 | 112 | 13 |  |  |     |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )   |
| PID        | Property identification number of a PGAP entry. ( $\text{Integer} > 0$ ; Default = EID)   |
| GA, GB     | Connected grid points at ends A and B. ( $\text{Integers} > 0$ ; $\text{GA} \neq \text{GB}$ )   |
| X1, X2, X3 | Components of the orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)   |
| GO         | Alternate method to supply the orientation vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. (Integer $> 0$ )  |
| CID        | Element coordinate system identification number. CID must be specified if GA and GB are coincident ( $\text{distance from GA to GB} < 10^{-4}$ ). See Remark 6. (Integer $\geq 0$ or blank) |

**Remarks:**

1. The CGAP element is intended for the nonlinear solution sequences 106, 129, 153, 159, and 400. However, it will produce a linear stiffness matrix for the other solutions, but remains linear with the initial stiffness. The stiffness used depends on the value for the initial gap opening (U0 field in the PGAP entry).
2. The gap element coordinate system is defined by one of two following methods:
  - If the coordinate system (CID field) is specified, the element coordinate system is established using that coordinate system, in which the element x-axis is in the T1 direction and the y-axis in the T2 direction. The orientation vector  $\vec{v}$  will be ignored in this case.
  - If the CID field is blank and the grid points GA and GB are not coincident (distance from A to B  $\geq 10^{-4}$ ), then the line AB is the element x-axis and the orientation vector  $\vec{v}$  lies in the x-y plane (like the CBEAM element).

3. The element coordinate system does not rotate as a result of deflections.
4. Initial gap openings are specified on the PGAP entry and not derived from the separation distance between GA and GB.
5. Forces are requested with the FORCE or NLSTRESS Case Control command. The NLSTRESS command is only for nonlinear solutions, and the output also includes the gap STATUS. Forces are output in the element coordinate system. The force  $F_x$  is positive for compression.
6. If CID is being used to define the element coordinate system and the CID refers to either a cylindrical or spherical coordinate system then grid GA will be used to locate the system. If grid GA lies on the z-axis of the cylindrical or spherical coordinate system it is recommended that a different coordinate system be used for this element.
7. See PARAM,CDITER for an alternative approach.

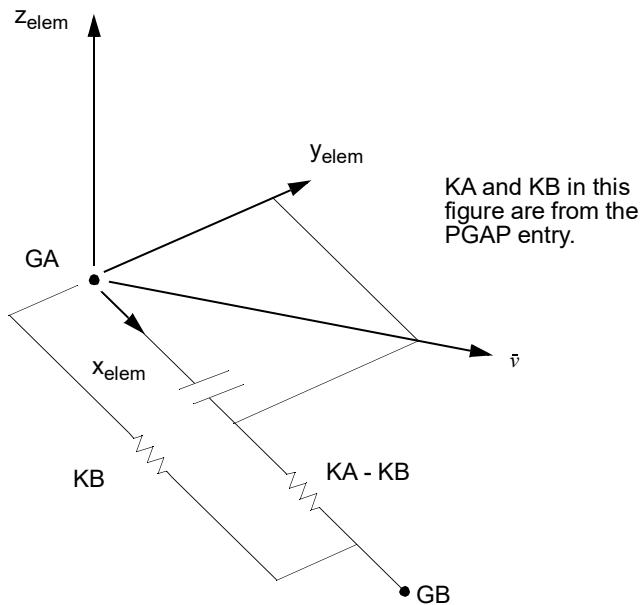


Figure 9-27 CGAP Element Coordinate System

8. Since a large stiffness is used for KA (the closed GAP stiffness), param,g damping should be avoided. Instead damping should be specified on the MATi entries and PARAM,W4 set.

**CHACAB****Acoustic Absorber Element Connection**

Defines the acoustic absorber element in coupled fluid-structural analysis.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8  | 9  | 10 |
|--------|-----|-----|-----|-----|-----|-----|----|----|----|
| CHACAB | EID | PID | G1  | G2  | G3  | G4  | G5 | G6 |    |
|        | G7  | G8  | G9  | G10 | G11 | G12 |    |    |    |
|        |     |     | G17 | G18 | G19 | G20 |    |    |    |

**Example:**

|        |    |    |   |   |   |   |   |   |  |
|--------|----|----|---|---|---|---|---|---|--|
| CHACAB | 95 | 12 | 1 | 2 | 5 | 7 | 8 | 9 |  |
|        | 24 | 23 |   |   |   |   |   |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )                       |
| PID        | Property identification number of a PACABS entry. ( $\text{Integer} > 0$ )                  |
| Gi         | Grid point identification numbers of connection points. ( $\text{Integer} \geq 0$ or blank) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grid points G1 through G4 and G9 through G12 is assumed to be in contact with the structure.

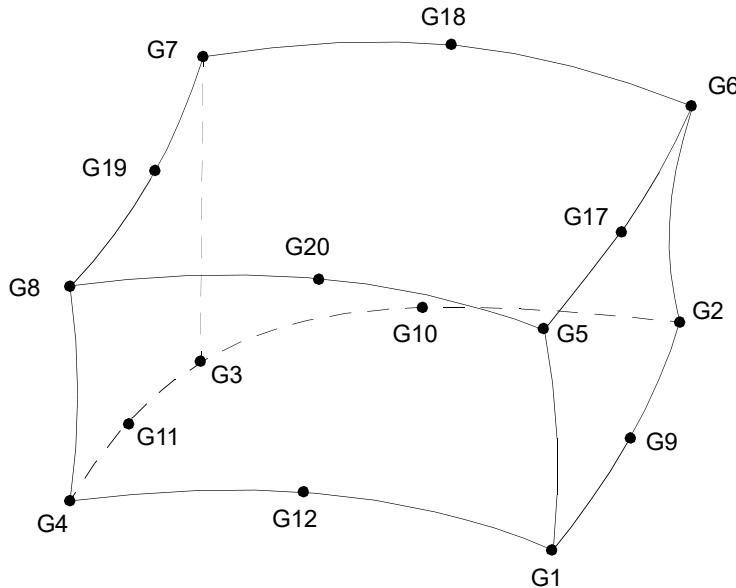


Figure 9-28 CHACAB Element Connection

6. The mass is lumped to the face formed by grid points G5 through G8 and G17 through G20 and defined to be in contact with the fluid. The opposite face has no mass contribution due to the absorber element. Also, the face in contact with the fluid has only translational stiffness in the direction normal to the face.

**CHACBR****Acoustic Barrier Element Connection**

Defines the acoustic barrier element.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8  | 9  | 10 |
|--------|-----|-----|-----|-----|-----|-----|----|----|----|
| CHACBR | EID | PID | G1  | G2  | G3  | G4  | G5 | G6 |    |
|        | G7  | G8  | G9  | G10 | G11 | G12 |    |    |    |
|        |     |     | G17 | G18 | G19 | G20 |    |    |    |

**Example:**

|        |    |    |   |   |   |   |   |   |  |
|--------|----|----|---|---|---|---|---|---|--|
| CHACBR | 95 | 12 | 1 | 2 | 5 | 7 | 8 | 9 |  |
|        | 24 | 23 |   |   |   |   |   |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)            |
| PID        | Property identification number of a PACBAR entry. (Integer > 0)       |
| Gi         | Grid point identification numbers of connection points. (Integer > 0) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grids G1 through G4 and G9 through G12 is assumed to be the backing that corresponds to MBACK on the PACBAR entry.
6. The face consisting of grid points G5 through G8 and G17 through G20 is assumed to be the septum that corresponds to MSEPTM on the PACBAR entry.

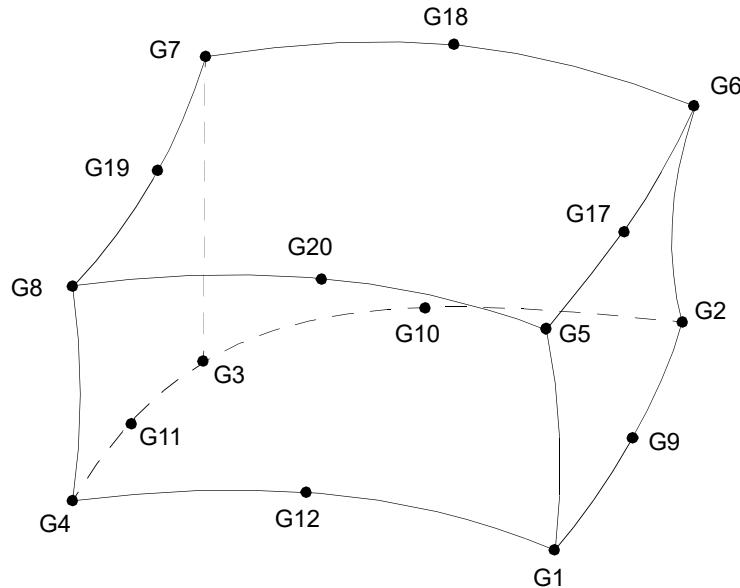


Figure 9-29 CHACBR Element Connection

7. The face in contact with the fluid is defined to be the face formed by grid points G5 through G8 and G17 through G20 and has only translational stiffness in the direction normal to the face.

**CHBDYE**

## Geometric Surface Element Definition (Element Form)

Defines a boundary condition surface element with reference to a heat conduction element.

**Format:**

| 1      | 2   | 3    | 4    | 5      | 6      | 7       | 8       | 9 | 10 |
|--------|-----|------|------|--------|--------|---------|---------|---|----|
| CHBDYE | EID | EID2 | SIDE | IVIEWF | IVIEWB | RADMIDF | RADMIDB |   |    |

**Example:**

|        |   |    |   |   |   |   |   |  |  |
|--------|---|----|---|---|---|---|---|--|--|
| CHBDYE | 2 | 10 | 1 | 3 | 3 | 2 | 2 |  |  |
|--------|---|----|---|---|---|---|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Surface element identification number for a specific side of a particular element. See Remarks 1. and 9. (Unique ( $0 < \text{Integer} < 100,000,000$ ) among all elements.) |
| EID2       | A heat conduction element identification number. ( $\text{Integer} > 0$ )  |
| SIDE       | A consistent element side identification number. See Remark 6. ( $1 \leq \text{Integer} \leq 6$ )  |
| IVIEWF     | A VIEW entry identification number for the front face of surface element. See Remark 2. for default. ( $\text{Integer} \geq 0$ )   |
| IVIEWB     | A VIEW entry identification number for the back face of surface element. See Remark 2. for default. ( $\text{Integer} \geq 0$ )  |
| RADMIDF    | RADM identification number for front face of surface element. See Remark 2. for default. ( $\text{Integer} \geq 0$ )   |
| RADMIDB    | RADM identification number for back face of surface element. See Remark 2. for default. ( $\text{Integer} \geq 0$ )  |

**Remarks:**

1. EID is a unique elemental ID associated with a particular surface element. EID2 identifies the general heat conduction element being considered for this surface element.
2. The defaults for IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank both on the CHBDYE entry and the BDYOR entry, then the default is zero.
3. For the front face of shell elements, the right-hand rule is used as one progresses around the element surface from G1 to G2 to ... Gn. For the edges of shell elements or the ends of line elements, an outward normal is used to define the front surface.
4. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
5. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.

6. Side conventions for solid elements.

The sides of the solid elements are numbered consecutively according to the order of the grid point numbers on the solid element entry. The sides of solid elements are either quadrilaterals or triangles. For each element type, tabulate the grid points (gp) at the corners of each side.

**8-node or 20-node CHEXA**

| side | gp | gp | gp | gp |
|------|----|----|----|----|
| 1    | 4  | 3  | 2  | 1  |
| 2    | 1  | 2  | 6  | 5  |
| 3    | 2  | 3  | 7  | 6  |
| 4    | 3  | 4  | 8  | 7  |
| 5    | 4  | 1  | 5  | 8  |
| 6    | 5  | 6  | 7  | 8  |

**CPENTA**

| side | gp | gp | gp | gp |
|------|----|----|----|----|
| 1    | 3  | 2  | 1  |    |
| 2    | 1  | 2  | 5  | 4  |
| 3    | 2  | 3  | 6  | 5  |
| 4    | 3  | 1  | 4  | 6  |
| 5    | 4  | 5  | 6  |    |

**CTETRA**

| side | gp | gp | gp |
|------|----|----|----|
| 1    | 3  | 2  | 1  |
| 2    | 1  | 2  | 4  |
| 3    | 2  | 3  | 4  |
| 4    | 3  | 1  | 4  |

**CPYRAM**

| side | gp | gp | gp | gp |
|------|----|----|----|----|
| 1    | 3  | 2  | 5  |    |
| 2    | 1  | 4  | 5  |    |
| 3    | 2  | 1  | 5  |    |
| 4    | 3  | 5  | 4  |    |
| 5    | 4  | 1  | 2  | 3  |

## 7. Side conventions for shell elements.

Side 1 of shell elements (top) are of an AREA type, and additional sides (2 through a maximum of 5 for a QUAD) are of LINE type. (See [CHBDYG, 1525](#) for surface type definition.)

Area Type Sides -- The first side is that given by the right-hand rule on the shell elements grid points.

Line Type Sides -- The second side (first line) proceeds from grid point 1 to grid point 2 of the shell element, and the remaining lines are numbered consecutively. The thickness of the line is that of the shell element, and the normal to the line is outward from the shell element in the plane of the shell. Note that any midside nodes are ignored in this specification.

For 3-D heat shell elements when used in combination with linear or quadratic nodal temperature (see option TEMPP of the NLMOPTS entry), SIDE=6 refers to BOT surface. SIDE=1 refers to TOP surface. The IVIEW and RADM should be filled in consistently in relation with the SIDE entry. E.g., when SIDE=1 or 6 then only IVIEWF makes sense.

## 8. Side conventions for line elements.

LINE elements have one linear side (side 1) with geometry that is the same as that of the element and two POINT-type sides corresponding to the two points bounding the linear element (first grid point-side 2; second grid point-side 3).

The TUBE-type element has two linear sides of type TUBE. The first side represents the outside with diameters equal to that of the outside of the tube. The second side represents the inside with diameters equal to that of the inside of the tube.

Point Sides -- Point sides may be used with any linear element. The direction of the outward normals of these points is in line with the element axis, but pointing away from the element. The area assigned to these POINT-type sides is consistent with the element geometry.

Rev Sides -- The CTRIA6 element has associated with it three REV sides. The first side is associated with Grid Points G1, G2, and G3. The positive face identification normals point away from the element.

9. Application of boundary conditions to CHBDYE is referenced through the EID. Boundary conditions can reference either the front or back face of the CHBDYE by specifying +EID or -EID respectively. Correspondingly, the back face is minus the normal vector of the front face. Similarly, IVIEWF and RADMIDF are associated with +EID and IVIEWB and RADMIDB with -EID. For radiation problems, if the RADMIDF or RADMIDB is zero, default radiant properties assume perfect black body behavior.
10. Starting with MSC Nastran 2004, axisymmetric view factors are supported CHBDYG of TYPE=REV, but not supported CHBDYE. If CHBDYE is used for this, axisymmetric view factors are not calculated.

**CHBDYG**

## Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

**Format:**

| 1      | 2   | 3  | 4    | 5      | 6      | 7       | 8       | 9  | 10 |
|--------|-----|----|------|--------|--------|---------|---------|----|----|
| CHBDYG | EID |    | TYPE | IVIEWF | IVIEWB | RADMIDF | RADMIDB |    |    |
|        | G1  | G2 | G3   | G4     | G5     | G6      | G7      | G8 |    |

**Example:**

|        |     |     |       |     |   |   |   |  |  |
|--------|-----|-----|-------|-----|---|---|---|--|--|
| CHBDYG | 2   |     | AREA4 | 3   | 3 | 2 | 2 |  |  |
|        | 100 | 103 | 102   | 101 |   |   |   |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Surface element identification number. (Unique (0 < Integer < 100,000,000) among all elemental entries)      |
| TYPE       | Surface type. See Remark 3. (Character)  |
| IVIEWF     | A VIEW entry identification number for the front face. See Remark 2. for default. (Integer $\geq 0$ )        |
| IVIEWB     | A VIEW entry identification number for the back face. See Remark 2. for default. (Integer $\geq 0$ )         |
| RADMIDF    | RADM identification number for front face of surface element. See Remark 2. for default. (Integer $\geq 0$ ) |
| RADMIDB    | RADM identification number for back face of surface element. See Remark 2. for default. (Integer $\geq 0$ )  |
| Gi         | Grid point IDs of grids bounding the surface. (Integer $> 0$ )   |

**Remarks:**

1. EID is a unique ID associated with a particular surface element as defined by the grid points.
2. The defaults for TYPE, IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYG entry and the BDYOR entry, then the default is zero.
3. TYPE specifies the kind of element surface; allowed types are: REV, REV1, AREA3, AREA4, AREA6, and AREA8. See [Figure 9-30](#), [Figure 9-31](#), and [Figure 9-32](#).
  - TYPE = REV

The “REV” type has two primary grid points that must lie in the x-z plane of the basic coordinate system with  $x > 0$ . A midside grid point G3 is optional and supports convection or heat flux from the edge of the six-noded CTRIA6 element. The defined area is a conical section with z as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.

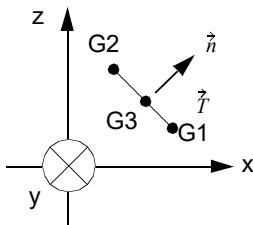


Figure 9-30 Normal Vector for CHBDYG Element of Type “REV”

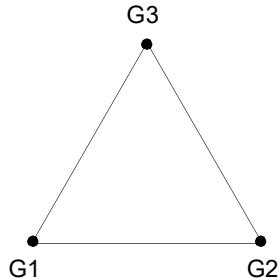
The unit normal lies in the x-z plane, and is given by

$$\hat{n} = (\hat{e}_y \times \vec{T}) / |\hat{e}_y \times \vec{T}| .$$

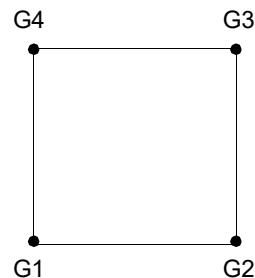
$\hat{e}_y$  is the unit vector in the y direction.

- TYPE = REV1. The “REV1” type has two primary grid points that must lie in the x-y plane of the basic coordinate system with  $x > 0$ . A midside grid point G3 is optional and supports convection or heat flux from the edge of the CQUADX (4 or 8 node) and CTRIA6 (3 or 6 node) element. The defined area is a conical section with y as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.
- TYPE = AREA3, AREA4, AREA6, or AREA8

These types have three and four primary grid points, respectively, that define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property entry is required for convection, radiation, or thermal vector flux.



AREA3



AREA4

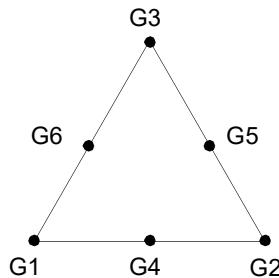
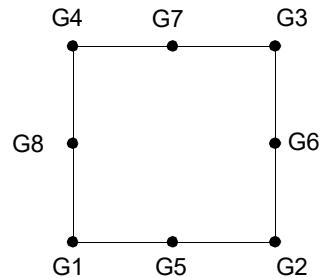
AREA6  
(Grid points G4 through G6 optional)AREA8  
(Grid points G5 through G8 optional)

Figure 9-31 TYPE Examples

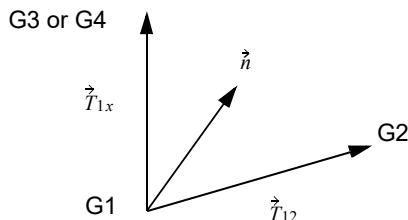


Figure 9-32 Normal Vector for CHBDYG Element of Types "AREAI"

The unit normal vector is given by

$$\hat{n} = \frac{(\vec{T}_{12} \times \vec{T}_{1x})}{|\vec{T}_{12} \times \vec{T}_{1x}|}$$

(G3 is used for triangles, and G4 is used for quadrilaterals.)

4. For defining the front face, the right-hand rule is used on the sequence G1 to G2 to ... Gn of grid points.

5. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
6. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.
7. See Remark 9. of CHBDYE for application of boundary conditions using CHBDYG entries and a discussion of front and back faces.

**CHBDYP**

## Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

**Format:**

| 1      | 2       | 3       | 4    | 5      | 6      | 7  | 8  | 9  | 10 |
|--------|---------|---------|------|--------|--------|----|----|----|----|
| CHBDYP | EID     | PID     | TYPE | IVIEWF | IVIEWB | G1 | G2 | G0 |    |
|        | RADMIDF | RADMIDB | GMID | CE     | E1     | E2 | E3 |    |    |

**Example:**

|        |   |   |       |   |     |     |     |     |  |
|--------|---|---|-------|---|-----|-----|-----|-----|--|
| CHBDYP | 2 | 5 | POINT | 2 | 2   | 101 |     | 500 |  |
|        | 3 | 3 |       |   | 0.0 | 0.0 | 1.0 |     |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Surface element identification number. (Unique $0 < \text{Integer} < 100,000,000$ among all element identification numbers.)         |
| PID        | PHBDY property entry identification numbers. ( $\text{Integer} > 0$ )  |
| TYPE       | Surface type. See Remark 3. (Character)  |
| IVIEWF     | VIEW entry identification number for the front face. ( $\text{Integer} \geq 0$ or blank)   |
| IVIEWB     | VIEW entry identification number for the back face. ( $\text{Integer} \geq 0$ or blank)  |
| G1, G2     | Grid point identification numbers of grids bounding the surface. ( $\text{Integer} > 0$ )  |
| GO         | Orientation grid point. ( $\text{Integer} \geq 0$ ; Default = 0)   |
| RADMIDF    | RADM entry identification number for front face. ( $\text{Integer} \geq 0$ or blank)   |
| RADMIDB    | RADM entry identification number for back face. ( $\text{Integer} \geq 0$ or blank)  |
| GMID       | Grid point identification number of a midside node if it is used with the line type surface element.                                 |
| CE         | Coordinate system for defining orientation vector. ( $\text{Integer} \geq 0$ ; Default = 0)  |
| Ei         | Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank) |

**Remarks:**

1. EID is a unique ID associated with a particular surface element as defined by the grid point(s).
2. The defaults for PID, TYPE, IVIEWF, IVIEWB, GO, RADMIDF, RADMIDB, CE, and Ei may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYP entry and the BDYOR entry, then the default is zero.

3. TYPE specifies the kind of element surface; the allowed types are: "POINT," "LINE," "ELCYL," "FTUBE," and "TUBE." For TYPE = "FTUBE" and TYPE = "TUBE," the geometric orientation is completely determined by G1 and G2; the GO, CE, E1, E2, and E3 fields are ignored.

- TYPE = "POINT"

TYPE = "POINT" has one primary grid point, requires a property entry, and the normal vector  $\vec{n}$  must be specified if thermal flux is to be used.

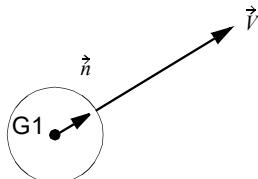


Figure 9-33 Normal Vector for CHBDYP Element of Type "POINT"  
(See Remarks 4. and 5.)

The unit normal vector is given by  $\hat{n} = \vec{V}/|\vec{V}|$  where  $\vec{V}$  is specified in the Ei field and given in the basic system at the referenced grid point. See Remarks 4. and 5. for the determination of  $\vec{V}$ .

- TYPE = "LINE," "FTUBE," or "TUBE"

The TYPE = "LINE" type has two primary grid points, requires a property entry, and the vector is required. TYPE = "FTUBE" and TYPE = "TUBE" are similar to TYPE = "LINE" except they can have linear taper with no automatic view factor calculations. GMID is an option for the TYPE = "LINE" surface element only and is ignored for TYPE = "FTUBE" and "TUBE".

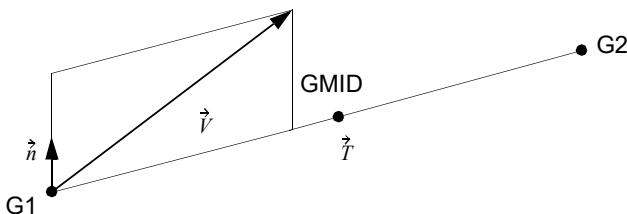


Figure 9-34 Normal Vector for CHBDYP Element with TYPE="LINE",  
TYPE="FTUBE", or TYPE="TUBE" (See Remarks 4. and 5.)

The unit normal lies in the plane  $\vec{V}$  and  $\vec{T}$ , is perpendicular to  $\vec{T}$ , and is given by:

$$\hat{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{|\vec{T} \times (\vec{V} \times \vec{T})|}$$

- TYPE = "ELCYL"

TYPE = "ELCYL" (elliptic cylinder) has two connected primary grid points and requires a property entry. The vector must be nonzero.

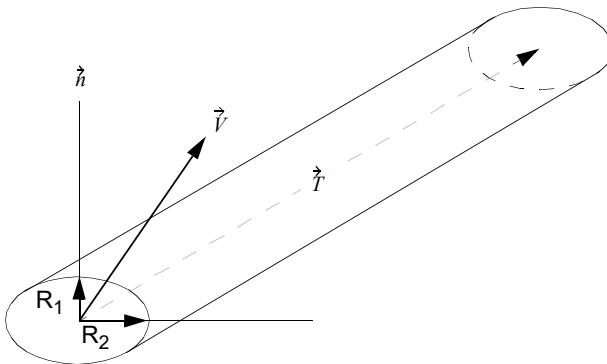


Figure 9-35 Normal Vector for CHBDYP Element of TYPE="ELCYL"  
(See Remarks 4. and 5.)

The same logic is used to determine  $\hat{n}$  as for TYPE = LINE. The "radius"  $R_1$  is in the  $\hat{n}$  direction, and  $R_2$  is the perpendicular to  $\hat{n}$  and  $\hat{T}$  (see fields 7 and 8 of PHBDY entry).

4. For TYPE = "POINT," TYPE = "LINE," and TYPE = "ELCYL," geometric orientation is required. The required information is sought in the following order:
  - If GO > 0 is found on the CHBDYP entry, it is used.
  - Otherwise, if a nonblank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
  - If neither of the above, the same information is sought in the same way from the BDYOR entry.
  - If none of the above apply, a warning message is issued.
5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.
  - If GO > zero:
 

For a TYPE = "POINT" surface, the normal to the front face is the vector from G1 to GO. For the TYPE = "LINE" surface, the plane passes through G1, G2, GO and the right-hand rule is used on this sequence to get the normal to the front face. For TYPE = "ELCYL" surface the first axis of the ellipse lies on the G1, G2, GO plane, and the second axis is normal to this plane. For TYPE = "FTUBE" or "TUBE" surface, no orientation is required, and GO is superfluous.
  - If GO is zero:
 

For a TYPE = "POINT" surface, the normal to the front face is the orientation vector. For the TYPE = "LINE" surface, the plane passes through G1, G2, and the orientation vector; the front face is based on the right-hand rule for the vectors G2-G1 and the orientation vector. For TYPE = "ELCYL" surface, the first axis of the ellipse lies on the G1, G2, orientation vector plane, and the second axis is normal to this plane.
6. The continuation entry is optional.
7. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.

8. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP entries.
9. For RC network solver in thermal analysis, the G0, GMID, CE, E1, C2 and E3 are ignored.

**CHEXA****Six-Sided Solid Element Connection**

Defines the connections of the six-sided solid element with eight to twenty grid points or the six-sided solid shell element with eight grid points.

**Format:**

| 1     | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CHEXA | EID | PID | G1  | G2  | G3  | G4  | G5  | G6  |    |
|       | G7  | G8  | G9  | G10 | G11 | G12 | G13 | G14 |    |
|       | G15 | G16 | G17 | G18 | G19 | G20 |     |     |    |

**Example:**

|       |    |    |    |    |    |    |    |    |  |
|-------|----|----|----|----|----|----|----|----|--|
| CHEXA | 71 | 4  | 3  | 4  | 5  | 6  | 7  | 8  |  |
|       | 9  | 10 | 0  | 0  | 30 | 31 | 53 | 54 |  |
|       | 55 | 56 | 57 | 58 | 59 | 60 |    |    |  |

| Descriptor | Meaning  | Type                      | Default  |
|------------|--|---------------------------|----------|
| EID        | Element identification number.<br>( $0 < \text{Integer} < 100,000,000$ )                 | $\text{Integer} > 0$      | Required |
| PID        | Property identification number of a PSOLID, PLSOLID or PCOMPLS entry.                    | $\text{Integer} > 0$      | Required |
| Gi         | Grid point identification numbers of connection points.<br><br>Integer $\geq 0$ or blank | Integer $\geq 0$ or blank | Required |

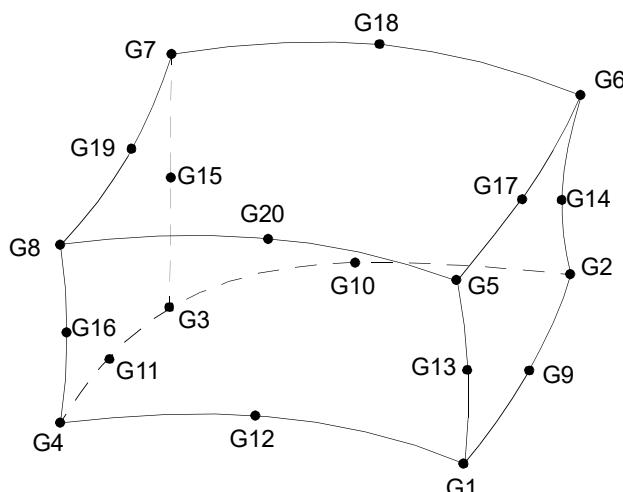


Figure 9-36      CHEXA Element Connection

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. For Nastran conventional element, the edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. Either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system. The material coordinate system is defined on the PSOLID entry.
5. The second continuation is optional.
6. For nonhyperelastic and non-composite elements, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.  
R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.  
S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.  
T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system a  $3 \times 3$  positive-definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

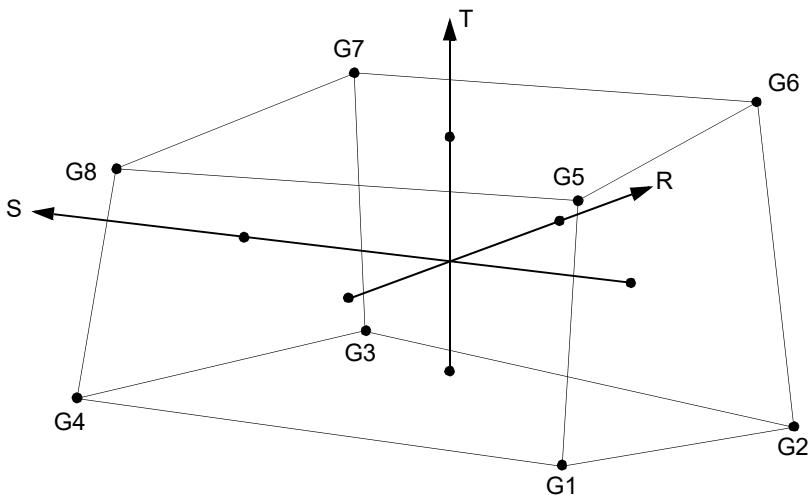


Figure 9-37 CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.
8. For hyperelastic elements, the plot codes are specified under the CHEXA FD element name in [Item Codes, 1045](#).
9. If a CHEXA element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
  - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
  - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
10. By default, all of the twelve edges of the element are considered straight unless:
  - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
  - For h-elements, any of G9 through G20 are specified.
11. This element may be used:
  - a. with BEH8=SLCOMP and INT8=ASTN, as a three-dimensional eight-node solid shell element in conjunction with PSLDN1 entry (SOL400 only),
  - b. with BEH8=SLCOMP and INT8=ASTN, as a three-dimensional eight-node composite solid shell element in conjunction with PCOMPLS entry with BEH8 = SLCOMP (INT8 = ASTN) (SOL 400 and all linear solution sequences between SOL101 and SOL112), or
  - c. with BEH8=SLCOMP and INT8=L or Q, or BEH20=SLCOMP and INT8=Q.
12. When this element is used as a three-dimensional eight-node solid shell element or three-dimensional eight-node composite solid shell element, the user should keep in mind that the layer orientation is required to be in the element T-direction, when specifying grid order.

13. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-38](#).

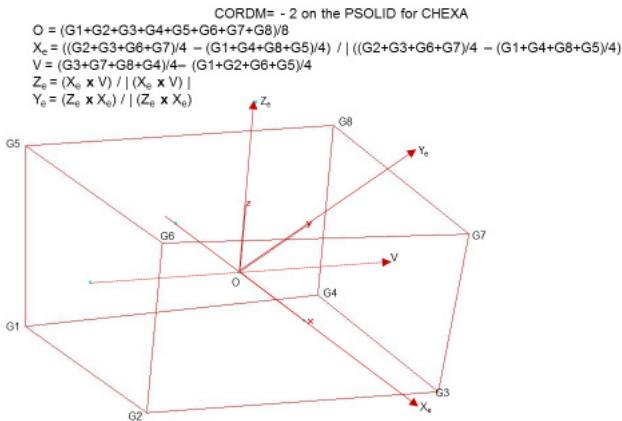


Figure 9-38      PSOLID for CHEXA

**CIFHEX****Solid InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, twenty-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CIFHEX | EID | PID | G1  | G2  | G3  | G4  | G5  | G6  |    |
|        | G7  | G8  | G9  | G10 | G11 | G12 | G13 | G14 |    |
|        | G15 | G16 | G17 | G18 | G19 | G20 |     |     |    |

**Example:**

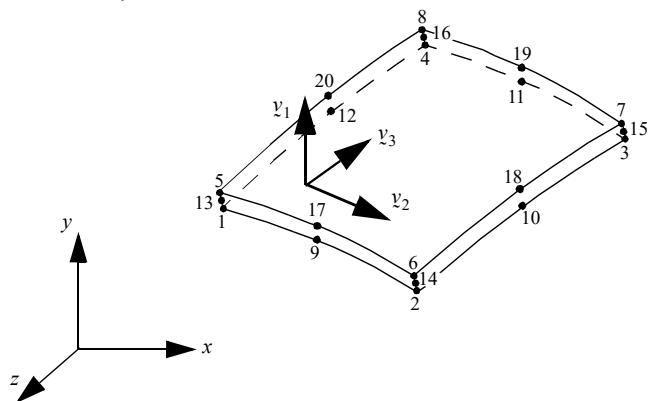
|        |     |     |     |     |     |     |     |     |  |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| CIFHEX | 700 | 701 | 456 | 357 | 882 | 889 | 443 | 447 |  |
|        | 162 | 911 |     |     |     |     |     |     |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property number of a PCOHE entry. (Integer > 0)  |
| G1-G8      | Identification number of connected corner grid points. Required data for all eight corner grid points. (Unique Integer > 0)  |
| G9-G12     | Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)   |
| G17-G20    | Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with twenty-noded hexahedral elements. (Unique Integer > 0) |
| G13-G16    |  |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure. Specify either G1-G8, or all G1-G20.
3. The element is typically used to model the interface between different materials, where G1, G9, G2, G10, G3, G11, G4 and G12 correspond to one side (called the bottom) and G5, G17, G6, G18, G7, G19, and G20 correspond to the other side (called the top). The stress components are one normal and two shear tractions. When only G1-G8 are specified, the element is linear. When in addition to G1-G8, G9-G12, G17-G20 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.

5. The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G4 and G5-G8 may coincide.



6. This element does not support thermal load. No thermal strain will be generated.

**CIFPENT****Solid InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, fifteen-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

**Format:**

| 1       | 2   | 3   | 4  | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|-----|-----|----|-----|-----|-----|-----|-----|----|
| CIFPENT | EID | PID | G1 | G2  | G3  | G4  | G5  | G6  |    |
|         | G7  | G8  | G9 | G10 | G11 | G12 | G13 | G14 |    |
|         | G15 |     |    |     |     |     |     |     |    |

**Example:**

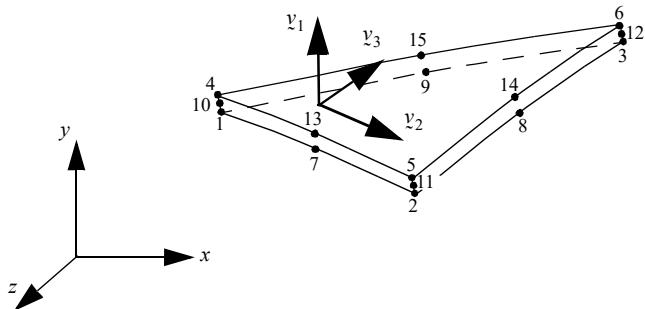
|         |     |     |     |     |     |     |     |     |  |
|---------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| CIFPENT | 700 | 701 | 456 | 357 | 882 | 889 | 443 | 447 |  |
|---------|-----|-----|-----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property number of a PCOHE entry. (Integer > 0)  |
| G1-G6      | Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)   |
| G7-G9      | Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)   |
| G13-G15    |  |
| G10-G12    | Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with fifteen-noded pentahedral elements. (Unique Integer > 0) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure. Specify either G1-G6 or all G1-G15.
3. The element is typically used to model the interface between different materials, where G1, G7, G2, G8, G3 and G9 correspond to one side (called the bottom) and G4, G13, G5, G14, G6, and G15 correspond to the other side (called the top). The stress components are one normal and one shear tractions. When only G1-G6 are specified, the element is linear. When in addition to G1-G6, G7-G9, G13-G15 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.

5. The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G3 and G4-G6 may coincide.



6. This element does not support thermal load. No thermal strain will be generated.

**CIFQDX****Axisymmetric InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, eight-node, axisymmetric interface element used to simulate the onset and progress of delamination in SOL 400.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|----|----|----|----|----|----|
| CIFQDX | EID | PID | G1 | G2 | G3 | G4 | G5 | G6 |    |
|        | G7  | G8  |    |    |    |    |    |    |    |

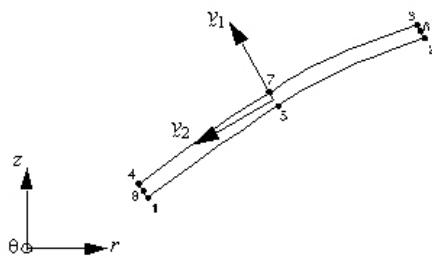
**Example:**

|        |      |     |     |     |     |     |     |     |  |
|--------|------|-----|-----|-----|-----|-----|-----|-----|--|
| CIFQDX | 700  | 701 | 456 | 357 | 882 | 889 | 443 | 447 |  |
|        | 1612 | 911 |     |     |     |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property number of a PCOHE entry. (Integer > 0)   |
| G1-G4      | Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)  |
| G5, G7     | Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)  |
| G6, G8     | Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral axisymmetric elements. (Unique Integer > 0) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where G1, G5, and G2 correspond to one side (called the bottom) and G3, G7, and G4 correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only G1-G4 are specified, the element is linear. When in addition to G1-G4, G5 and G7 are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges G1-G5-G2 and G3-G7-G4 may coincide.
6. The element must lie in the x-y plane of the basic system. Coordinate r is parallel to the x-basic and coordinate z is parallel to y-basic.



7. This element does not support thermal load. No thermal strain will be generated.

**CIFQUAD****Planar InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, eight-noded planar interface element used to simulate the onset and progress of delamination in SOL 400 only.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|-----|-----|----|----|----|----|----|----|----|
| CIFQUAD | EID | PID | G1 | G2 | G3 | G4 | G5 | G6 |    |
|         | G7  | G8  |    |    |    |    |    |    |    |

**Example:**

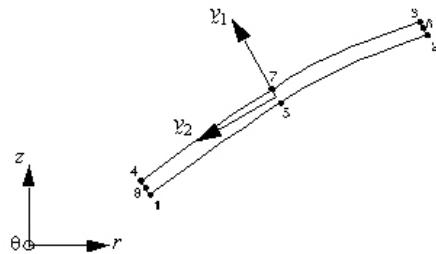
|         |      |     |     |     |     |     |     |     |  |
|---------|------|-----|-----|-----|-----|-----|-----|-----|--|
| CIFQUAD | 700  | 701 | 456 | 357 | 882 | 889 | 443 | 447 |  |
|         | 1612 | 911 |     |     |     |     |     |     |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property number of a PCOHE entry. (Integer > 0)  |
| G1-G4      | Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)   |
| G5, G7     | Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)   |
| G6, G8     | Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral elements. (Unique Integer > 0) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where G1, G5, and G2 correspond to one side (called the bottom) and G3, G7, and G4 correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only G1-G4 are specified, the element is linear. When in addition to G1-G4, G5 and G7 are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges G1-G5-G2 and G3-G7-G4 may coincide.

6. The element must lie in the x-y plane of the basic system.



7. This element does not support thermal load. No thermal strain will be generated.

**CINTC****Line Interface Element Connection**

Defines a line interface element with specified boundaries.

**Format:**

| 1   | 2   | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|-----|------|---|---|---|---|---|---|----|
| CINTC   | EID | TYPE |   |   |   |   |   |   |    |
| LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn)) |     |      |   |   |   |   |   |   |    |

**Example:**

|                                  |      |             |  |  |  |  |  |  |  |
|----------------------------------|------|-------------|--|--|--|--|--|--|--|
| CINTC                            | 1001 | GRDLIS<br>T |  |  |  |  |  |  |  |
| LIST=(101,102(Q),-103(Q),104(L)) |      |             |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| TYPE       | Connectivity. If TYPE = "GRDLIST" or blank (Default), the user will specify the boundaries via Bulk Data entry, GMBNDC. See Remark 2. (Character; Default = "GRDLIST") |
| BIDI       | Boundary curve identification number, referenced to Bulk Data entry, GMBNDC. See Remark 2. (Integer ≠ 0)   |
| INTPi      | Interpolation scheme. (Character; Default = "L")<br>INTP = "L": Linear interpolation;<br>INTP = "Q": Quadratic interpolation.  |

**Remarks:**

1. Line interface element identification numbers must be unique with respect to all other line interface elements.
2. There must be at least two BIDI specified. If all BIDI are positive, by default, the degrees of freedom associated with the grids on the boundary represented by the first BID will be taken as the independent (n-set), and the degrees of freedom with the grids on the rest of boundaries are taken as the dependent (m-set). If there is a single negative BID, the degrees of freedom associated with the grids on the boundary represented by this BID will be taken as the independent (n-set), and the rest of the degrees of freedom with other boundaries are used as the dependent (m-set). If there are two or more negative BIDs, the degrees of freedom with the first negative one will be taken as the independent.
3. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
4. The m-set degrees of freedom specified on the boundary grids by this entry may not be specified by other entries that define mutually exclusive sets.

**CLOAD**

## Static Load Combination for Superelement Loads (Superposition)

Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOLs 106 or 153).

Format:

| 1     | 2   | 3    | 4      | 5    | 6  | 7    | 8  | 9    | 10 |
|-------|-----|------|--------|------|----|------|----|------|----|
| CLOAD | CID | S    | S1     | IDV1 | S2 | IDV2 | S3 | IDV3 |    |
|       | S4  | IDV4 | -etc.- |      |    |      |    |      |    |

Example:

|       |       |     |      |    |      |     |       |     |  |
|-------|-------|-----|------|----|------|-----|-------|-----|--|
| CLOAD | 25    | 1.0 | 25.0 | 10 | -1.0 | 101 | 2.2-1 | 604 |  |
|       | -62.0 | 62  |      |    |      |     |       |     |  |

| Descriptor | Meaning   |
|------------|---|
| CID        | Combination identification number. (Integer > 0)  |
| S          | Scale factor. (Real)  |
| Si         | Scale factors. (Real)   |
| IDVi       | Identification numbers of load vectors (EXCITEID of a selected LSEQ entry) calculated for a superelement loads entry. (Integer > 0) |

Remarks:

1. The CLOAD entry must be selected in the residual solution subcases of the Case Control with CLOAD = CID and must be used if loads are applied to upstream superelements in SOL 106 or 153.
2. The load vector defined is given by  $\{P\} = S \sum_i S_i \{P_{IDVi}\}$
3. The IDVi field refers to a previously calculated load vector for the superelement via the LSEQ approach. That is, a LOADSET keyword must have been selected in Case Control that in turn refers to one or more LSEQ entries in the Bulk Data Section. The IDVi refers to the EXCITEID of such LSEQ entries. For more details, see the Case Control commands [LSEQ, 2190](#) Bulk Data entry and the [LOADSET \(Case\), 439](#).
4. In the CID or IDV fields, a CLOAD entry may not reference an identification number defined by another CLOAD entry.

**CMARKB2****Two-Noded Marker Connectivity Definition**

Defines a 2-noded marker beam element by means of connecting two grid points. Used in SOL 700 only.

**Format:**

| 1       | 2  | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|----|-----|----|----|---|---|---|---|----|
| CMARKB2 | ID | PID | G1 | G2 |   |   |   |   |    |

**Example:**

|         |   |   |   |    |  |  |  |  |  |
|---------|---|---|---|----|--|--|--|--|--|
| CMARKB2 | 7 | 1 | 9 | 10 |  |  |  |  |  |
|---------|---|---|---|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique element number. (Integer > 0; Required)                    |
| PID        | Property ID referring to a PMARKER entry. (Integer > 0; Required) |
| G1         | Grid point number connectivity 1. (Integer > 0; Required)         |
| G2         | Grid point number connectivity 2. (Integer > 0; Required)         |

**Remarks:**

1. A CMARKB2 element may refer to two types of grid points:
  - a. Structural grid points, thus grid points that are part of the connectivity of an element
  - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.

**CMARKN1****One-Noded Marker Connectivity Definition**

Defines a 1-noded marker element on a grid point. Used in SOL 700 only.

Format:

| 1       | 2  | 3   | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|-----|----|---|---|---|---|---|----|
| CMARKN1 | ID | PID | G1 |   |   |   |   |   |    |

Example:

|         |   |   |   |  |  |  |  |  |  |
|---------|---|---|---|--|--|--|--|--|--|
| CMARKN1 | 7 | 1 | 9 |  |  |  |  |  |  |
|---------|---|---|---|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique element number. (Integer > 0; Required)                    |
| PID        | Property ID referring to a PMARKER entry. (Integer > 0; Required) |
| G          | Grid point number. (Integer > 0; Required)                        |

Remarks:

1. A CMARKB2 element may refer to two types of grid points:
  - a. Structural grid points, thus grid points that are part of the connectivity of an element
  - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.

**CMASS1**

## Scalar Mass Connection

Defines a scalar mass element.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|----|----|----|----|---|---|----|
| CMASS1 | EID | PID | G1 | C1 | G2 | C2 |   |   |    |

**Example:**

|        |    |   |   |   |  |  |  |  |  |
|--------|----|---|---|---|--|--|--|--|--|
| CMASS1 | 32 | 6 | 2 | 1 |  |  |  |  |  |
|--------|----|---|---|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )             |
| PID        | Property identification number of a PMASS entry. ( $\text{Integer} > 0$ ; Default = EID) |
| G1, G2     | Geometric grid or scalar point identification number. ( $\text{Integer} \geq 0$ )        |
| C1, C2     | Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point)      |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must not be coincident.
4. For a discussion of the scalar elements, see the *MSC Nastran Reference Guide*, Section 5.6.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Scalar elements input coupled mass matrices when the second pair of fields is entered. When uncoupled point masses are desired input only the first pair of fields. When a coupled mass matrix is requested the submatrix added has M on the diagonal, and -M on the off-diagonal. The element is not checked for internal constraints, which is the user's responsibility if desired. There are instances where elements with internal constraints are desired, although not frequently. To identify the presence of internal constraints caused by coupled mass, inspect GPWG output, OLOAD output due to GRAV loads, and rigid body modes of free structures. Some forms of coupled mass will cause coupling of rigid body translational mass terms in GPWG output, and poor rigid body modes in modal analysis.

**CMASS2**

## Scalar Mass Property and Connection

Defines a scalar mass element without reference to a property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|---|----|----|----|----|---|---|----|
| CMASS2 | EID | M | G1 | C1 | G2 | C2 |   |   |    |

**Example:**

|        |    |      |   |   |  |  |  |  |  |
|--------|----|------|---|---|--|--|--|--|--|
| CMASS2 | 32 | 9.25 | 6 | 1 |  |  |  |  |  |
|--------|----|------|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)             |
| M          | Value of the scalar mass. (Real)  |
| G1, G2     | Geometric grid or scalar point identification number. (Integer $\geq 0$ )     |
| C1, C2     | Component number. (0 $\leq$ Integer $\leq 6$ ; blank or zero if scalar point) |

**Remarks:**

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for Gi and Ci.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. See Remark 7 for [CMASS1, 1549](#).

**CMASS3****Scalar Mass Connection to Scalar Points Only**

Defines a scalar mass element that is connected only to scalar points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|----|----|---|---|---|---|----|
| CMASS3 | EID | PID | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |    |    |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|
| CMASS3 | 13 | 42 | 62 |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )             |
| PID        | Property identification number of a PMASS entry. ( $\text{Integer} > 0$ ; Default = EID) |
| S1, S2     | Scalar point identification numbers. ( $\text{Integer} \geq 0$ ; $S1 \neq S2$ )          |

**Remarks:**

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CMASS4****Scalar Mass Property and Connection to Scalar Points Only**

Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.

**Format:**

| 1      | 2   | 3 | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|----|----|---|---|---|---|----|
| CMASS4 | EID | M | S1 | S2 |   |   |   |   |    |

**Example:**

|        |    |       |  |    |  |  |  |  |  |
|--------|----|-------|--|----|--|--|--|--|--|
| CMASS4 | 23 | 14.92 |  | 23 |  |  |  |  |  |
|--------|----|-------|--|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)       |
| M          | Scalar mass value. (Real)   |
| S1, S2     | Scalar point identification numbers. (Integer $\geq 0$ ; S1 $\neq$ S2 ) |

**Remarks:**

1. S1 or S2 may be blank or zero, indicating a constrained coordinate. This is the usual case.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CMREBAI**

Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR with INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Unlike the CMREBAR element, this element can span multiple matrix CHEXA elements (for example, could be allied to all elements on the bottom of a flat surface modeled with several layers of CHEXA matrix elements through the thickness).

**Format:**

| 1       | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|----|----|----|----|----|----|----|----|
| CMREBAI | ID | IP | G1 | G2 | G3 | G4 | G5 | G6 |    |
|         | G7 | G8 |    |    |    |    |    |    |    |

**Example:**

|         |     |   |    |    |    |    |   |   |  |
|---------|-----|---|----|----|----|----|---|---|--|
| CMREBAI | 100 | 2 | 31 | 32 | 88 | 87 |   |   |  |
| CMREBAI | 250 | 8 | 1  | 2  | 3  | 4  | 5 | 6 |  |
|         | 7   | 8 |    |    |    |    |   |   |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)   |
| IP         | Property identification of a matching PMRBAR entry. (Integer; Required; no Default)   |
| G1-G4      | Grid point identification numbers of the four corner points. (Integer > 0; all unique)  |
| G5-G8      | Grid point identification numbers of the four mid-side nodes similar to G5-G8 for the CQUAD8 element. (Integer > 0, or blank. If any of G5-G8 are not blank, all nodes in the range G5 to G8 must be defined and must be unique among the range G1-G8.) |

**Remarks:**

1. This entry makes use of Marc's REBAR and INSERT capabilities for membrane element types 147 and 148. If G5-G8 are blank, Marc element 147 with 4 nodes is used. If G5-G8 are not blank, Marc element 148 with 4 corner nodes and 4 mid-side nodes is used. Entry CMREBAR makes use of Marc's REBAR capability (without INSERT) and uses rebar elements 23 and 146.
2. The grid ID's do not have to correspond to those of any matrix CHEXA element. The PMREABI entry is used to describe the matrix CHEXA elements that these rebar elements will be inserted into.
3. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforces composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea

of rebar layer concept contains (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by embedding membrane rebar elements into solid matrix elements using Marc's INSERT option. Membrane rebar elements types 147 and 148 are available with this option. They are empty 4-node or 8-node quadrilaterals. You can place reinforcing cord layers within these empty elements. These elements are then embedded into their corresponding solid elements representing the matrix materials. Independent meshes can be used for the rebar membrane elements and the matrix elements. Marc's INSERT option is automatically invoked by the CMREBAI elements and used to enforce the compatibility between two different meshes.

4. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSERT option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
5. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
6. See MPREBAI for additional information and figures defining these rebar elements.
7. Only CHEXA elements may be used for the matrix elements.

**CMREBAR**

Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR without INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. CMREBAR elements require that the rebar be placed in matching CHEXA matrix elements on a one-to-one basis. For a similar capability where the rebar can span multiple CEHXA matrix elements, see the CMREBAI entry.

**Format:**

| 1       | 2  | 3  | 4   | 5   | 6    | 7 | 8 | 9 | 10 |
|---------|----|----|-----|-----|------|---|---|---|----|
| CMREBAR | ID | IP | ID2 | IDD | ID22 |   |   |   |    |

**Example:**

|         |      |    |     |      |     |  |  |  |  |
|---------|------|----|-----|------|-----|--|--|--|--|
| CMREBAR | 100  | 2  | 1   |      |     |  |  |  |  |
| CMREBAR | 1001 | 50 | 101 | 1100 | 200 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)  |
| IP         | Property identification of a matching PMRBAR entry. (Integer; Required; no Default)  |
| ID2        | CHEXA (8 node or 20 node) "matrix" element that the rebar will be added to. (Integer; Required; no Default)  |
| IDD        | If more than one rebar element in a continuous range is to be added to a continuous range of rebar elements, IDD represents the first rebar element identification number in the range. (Integer; Required; Default = 0, IDD must be larger than ID)             |
| ID22       | If more than one rebar element in a continuous range is to be added to a continuous range of "matrix" elements, ID22 represents the last CHEXA matrix element identification number in the range. (Integer; Required; Default = 0, ID22 must be larger than ID2) |

**Remarks:**

1. This entry makes use of Marc's REBAR capability for element types 23 and 146. Entry CMREBAI makes use of Marc's REBAR and INSERT capabilities and uses membrane rebar elements 147 and 148.
2. If IDD is entered, ID22 must also be entered and the difference IDD-ID must match the difference ID22-ID2.

3. The same grid ID's are used to define element ID and ID2. Similarly the same grid ID's define the other elements if a range of elements are used. Element ID is an "empty shell element" and up to 5 rebar layers (each containing multiple rebar) are placed in the empty shell as specified by the PMREBAR property entries.
4. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforces composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea of rebar layer concept contains that (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by superimposing solid rebar elements on corresponding solid matrix elements using the same element connectivity. The rebar elements are empty 8-node or 20-node CHEXA elements derived from the matching matrix elements. The reinforcing cord layers are placed within the elements. Each solid rebar element is then superimposed on a solid matrix element. The two elements share the same space with the same element connectivity (therefore, the same element nodes). The compatibility condition between the reinforcements and the matrix materials is then automatically enforced.
5. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSER T option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
6. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
7. See PMREBAR for additional information and figures defining these rebar elements.
8. Only CHEXA elements may be used for the matrix elements.

## Entries CO - CY

### COHESIV

Defines Data for Cohesive Materials in SOL 600

This option allows you to define material properties for interface elements, that may be used to simulate the onset or progress of delamination, and to associate these material properties with a list of element numbers. The cohesive material is defined using the cohesive energy (also called critical energy release rate), that equals the area below the equivalent traction versus equivalent relative displacement curve. The shape of this curve can be bilinear, exponential, or combined linear-exponential. Mixed mode delamination is incorporated by converting the normal and shear components of the relative displacements into an equivalent using the normal shear weighting factor.

As an alternative to the standard linear, exponential, and liner-exponential model, the user can also utilize this option to trigger the call to the UCOHESIVE user-subroutine. Used in SOL 600 only.

**Format:**

| 1       | 2    | 3     | 4     | 5    | 6   | 7     | 8 | 9 | 10 |
|---------|------|-------|-------|------|-----|-------|---|---|----|
| COHESIV | MID  | ITYPE | IACT  | NAME |     |       |   |   |    |
|         | EN   | COD   | MOD   | NSW  | SNW | DECAY |   | ' |    |
|         | VISC | RATE  | STIFF |      |     |       |   |   |    |
|         | ISET |       |       |      |     |       |   |   |    |

**Example:**

|         |       |     |      |          |          |    |  |  |  |
|---------|-------|-----|------|----------|----------|----|--|--|--|
| COHESIV | 101   | 1   | 1    | material | number10 | 1  |  |  |  |
|         | 2000. | .01 | .015 | 23.      | 24.      | 1. |  |  |  |
|         | .05   | 0.0 | 1.0  |          |          |    |  |  |  |
|         | 101   |     |      |          |          |    |  |  |  |

| Descriptor  | Meaning  |
|-------------|--|
| MID (3,1)   | Material ID - Must match a MATXXX entry. (Integer; no Default)   |
| ITYPE (3,2) | Type of cohesive model. (Integer > 0; Default = 1) <ul style="list-style-type: none"> <li>1 Bilinear model (Default)</li> <li>2 Exponential modes</li> <li>3 Combined linear-exponential model</li> <li>-1 User-defined using user-subroutine UCOHESIVE</li> </ul>   |
| IACT (3,3)  | Option to deactivate elements and output to t16 file. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0 Elements remain active regardless of the damage level.</li> <li>1 Deactivate the elements if the maximum damage in all the element integration points has been reached. Do not remove the elements from the t16 file.</li> </ul> |

| Descriptor  | Meaning  |
|-------------|--|
| 2           | Deactivate the elements if the maximum damage in all the element integration points has been reached. Remove the elements from the t16 file. |
| NAME (3,5)  | Name of the material. (Character up to 40 characters; no Default; optional entry)  |
| EN (4,1)    | Cohesive energy. (Real; no Default; Required value)  |
| COD (4,2)   | Critical opening displacement. (Real; no Default; Required value)  |
| MOD (4,3)   | Maximum opening displacement (Bilinear model only). (Real; no Default)   |
| NSW (4,4)   | Normal-shear weighting factor, beta. (Real; no Default; Required value)  |
| SNW (4,5)   | Shear-normal weighting factor. (Real; Default = 1.0)   |
| DECAY (4,6) | Exponential decay factor. (Linear-Exponential model only).(Real; Default = 1.0)  |
| VISC (4,7)  | Viscous energy dissipation factor (zero implies no viscous energy dissipation). (Real; Default = 0.0)  |
| RATE (4,8)  | Relative displacement reference rate. Only used if viscous energy dissipation is by setting VISC to a non-zero value. (Real; Default = 0.0)  |
| STIFF (4,9) | Stiffening factor in compression. (Real; Default = 1.0)  |
| ISET (5)    | ID of a SET3 entry defining the elements associated with this cohesive material. (Integer, no Default, Required value)                       |

### Remarks:

1. Values in parenthesis (i,j) refer to Marc's COHESIVE entry. (Datablock, field)
2. For solid elements, this entry may be used to add a "layer" or interface between the solid elements (CHEXA, CTETRA or CPENTA). This interface can fail or delaminate depending on the properties entered. It is used in conjunction with Marc element types 188, 192 and 193 and provides the material properties for these element types.
3. For shell elements (CQUADi, CTRIAi) this entry may be used to add a layer between the edges of adjacent shells. It is used in conjunction with Marc element types 186 and 187 and provides the material properties for these element types.
4. Solid or shell elements with a COHESIV MID will automatically be assigned Marc element types 186, 187, 188, 189, 192, 193 as appropriate to the type and number of grids defined for that element. These solid elements must either only have corner nodes or have the full parabolic number of nodes (for example, CHEXA must either have 8 nodes or 20 nodes).
5. For SOL 600, cohesive behavior is not available for axisymmetric or plane strain analyses.
6. MID must not be used by any other material such as MAT1, MAT2, etc.
7. All continuation lines are required.

**COHFRIC****Cohesive Friction**

Allows friction and sticking during tensile conditions at the coupling surface. Use SOL700 only.

**Format:**

|         |     |         |      |        |  |  |  |  |
|---------|-----|---------|------|--------|--|--|--|--|
| COHFRIC | CID | MAXSTRS | FRIC | REFVEL |  |  |  |  |
|---------|-----|---------|------|--------|--|--|--|--|

**Example:**

|         |     |       |      |   |  |  |  |  |
|---------|-----|-------|------|---|--|--|--|--|
| COHFRIC | 112 | 8e+10 | 8e+5 | 2 |  |  |  |  |
|---------|-----|-------|------|---|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CID        | Unique number of a COHFRIC entry. (Integer>0; required)  |
| MAXSTRS    | Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real $\geq 0.0$ ; default=0.0) |
| FRIC       | Friction stress under tensile conditions. (Real $\geq 0.0$ ; default=0.0)  |
| REFVEL     | Reference value for velocity. (Real $\geq 0.0$ ; default=0.0)  |

**Remarks:**

1. If the cohesive friction parameters is uniform across the coupling surface then DYPARAM, COHESION can be used instead.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals  $Fric \times \min\left(1, \frac{V_{REL, \text{tangential}}}{REFVEL}\right)$ . This is a viscous-like friction law.
3. This shear force opposes the relative tangential movement along the coupling surface.

**COMPUDS**

Orthotropic Failure Model that allows to model Property Degradation

Defines an orthotropic failure model for shell composites specified by a user subroutine. Used in SOL700 only.

**Format:**

| 1       | 2     | 3     | 4     | 5       | 6     | 7     | 8     | 9 | 10 |
|---------|-------|-------|-------|---------|-------|-------|-------|---|----|
| COMPUDS | MID   | GROUP | UNAME |         |       |       |       |   | +  |
| +       | E1    | E2    | N12   | G12     | G1,Z  | G2,Z  | RHO   |   | +  |
| +       | NV    | S     | ALPHA | TRSFAIL | F12   |       |       |   | +  |
| +       | XT    | XT    | YT    | YC      | PFD   | VALUE | PFDST |   | +  |
| +       | FBTEN | FBCOM | MXTEN | MXCOM   | MXSHR |       |       |   | +  |
| +       | PRDFT | PRDFC | PRDMT | PRDMC   | PRDSH |       |       |   | +  |
|         |       |       |       |         |       |       |       |   |    |
|         |       |       |       |         |       |       |       |   |    |
|         |       |       |       |         |       |       |       |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE excomp1 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

|         |      |         |     |       |       |       |       |  |   |
|---------|------|---------|-----|-------|-------|-------|-------|--|---|
| COMPUDS | 1    | EXCOMP1 |     |       |       |       |       |  | + |
| +       | 30E6 | 1.0E6   | 0.3 | 2.0E6 | 3.0E6 | 1.5E6 | 0.056 |  | + |
| +       | 5    | 100     |     |       |       |       |       |  |   |
| +       | 200  | 150     | 100 | 110   |       |       |       |  |   |

| Field | Contents   |
|-------|--|
| MID   | Unique material identification number. (Integer > 0; required)   |
| GROUP | The group name used for the FMS section CONNECT SERVICE statement. (Character; required).                                  |
| UNAME | User subroutine name associated with the entry. (Character; default=blank).  |
| E1    | Modulus of elasticity in longitudinal direction (also defined as fiber direction or one-direction). (Real > 0.0; required) |
| E2    | Modulus of elasticity in lateral direction (also defined as matrix direction or two-direction). (Real > 0.0; required)     |

| Field      | Contents   |
|------------|--|
| N12        | Poisson's ratio ( 2/ 1 for uniaxial loading in one-direction). Note that $21 = 1/ 2$ for uniaxial loading in two-direction is related to $12, E1, E2$ by the relation $12 E2 = 21 E1$ . (Real > 0.0; required) |
| G12        | In-plane shear modulus R. (Real > 0.0; required)   |
| G1,Z       | Transverse shear modulus for shear in 1-Z plane. (Real > 0.0; default= G12)  |
| RHO        | Mass density. (R > 0.0; required)  |
| NV         | Number of additional history variables for a user model. See Remark 8. (0 < Integer < 10; default=0)   |
| S          | Failure stress for in-plane shear. (Real $\geq$ 0.0; default=0.0)  |
| ALPHA      | Nonlinear shear coefficient. (Real $\geq$ 0.0; default=0.0)  |
| TRSFAIL    | Transverse shear failure. (Character; default=SUBL)<br><br>ELEM: Failure if element fails<br><br>SUBL: Failure if sublayer fails   |
| F12        | Interaction term in Tsai-Wu theory. (Real; default=0.0)  |
| XT, XC     | Tensile compressive failure stress in the large structural direction. (Real > 0.0; required)   |
| YT, YC     | Tensile compressive failure stress in the lateral direction. (Real > 0.0; required)  |
| PFD        | Post-failure degradation model. (Character; default=STEPS)<br><br>STEPS: Degrade stresses by time steps<br><br>TIME: Degrade stresses by time<br><br>VELOC: Degrade stresses by velocity                       |
| VALUE      | Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Integer or Real; default=100)   |
| PFDS       | Post-failure degradation start. (Character; default=INDV)<br><br>INDV Stresses are degraded per distinct failure mode.<br><br>ALL Stresses are degraded if all elastic constants are zero.                     |
| FBTEN,FBCO | Failure modes in fiber, matrix direction, and theory failure. (Character; default=blank)   |
| N,MXTEN,M  |  |
| XCOM,MXS   |  |
| HR         |  |

| Field | Contents  |
|-------|---|
| PRDFT | Property degradation due to fiber-tension failure. (Integer; default=1111)      |
| PRDFC | Property degradation due to fiber-compression failure. (Integer; default=1010)  |
| PRDMT | Property degradation due to matrix-tension failure. (Integer; default=0110)     |
| PRDMC | Property degradation due to matrix-compression failure. (Integer; default=0110) |
| PRDSH | Property degradation due to in-plane shear failure. (Integer; default=0001)     |

**Remarks:**

11. Please check MAT8 and MAT8A cards about the details of each field. The field definition is identical as MAT8 and MAT8A.
12. UNAME can be:

| Subroutine Name | Function   |
|-----------------|--|
| EXCOMP          | User defined orthotropic material models for shell composite |

**CONCTL**

Parameter SWLDPRM override for CFAST, CSEAM, and CWELD Connector Elements

This entry provides local connector search algorithm control to override SWLDPRM values.

Format:

| 1      | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| CONCTL | SET3ID |        | PARAM1 | VALUE1 | PARAM2 | VALUE2 | PARAM3 | VALUE3 |    |
|        | PARAM4 | VALUE5 | PARAM4 | VALUE5 | -etc.- |        |        |        |    |

Example:

|        |        |   |         |     |       |   |        |   |  |
|--------|--------|---|---------|-----|-------|---|--------|---|--|
| CONCTL | 75     |   | PROJTOL | 0.2 | PRTSW | 1 | GSMOVE | 3 |  |
|        | NREDIA | 2 |         |     |       |   |        |   |  |

|        |    |  |        |   |  |  |  |  |  |
|--------|----|--|--------|---|--|--|--|--|--|
| CONCTL | 83 |  | MOVGAB | 1 |  |  |  |  |  |
|--------|----|--|--------|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SET3ID     | The ID of a SET3 entry using (DES= ELEM) to identify a group of connector elements. (Integer>0)   |
| PARAMi     | Name of the connector parameter. Allowable names are listed in Table 8-62 of SWLDPRM. (Character) |
| VALUEi     | Value of the parameter. See Table 8-62 of SWLDPRM (Real or Integer)                               |

Remarks:

1. Multiple CONCTL entries are allowed and ALL appearing will be used. If the user supplies a PARAMi, VALUEi that refers to specific Element on one CONCTL entry and a same PARAMi with VALUEj that refers to the same specific Element on another CONCTL, a fatal message will be issued.
2. For a given connector element ID, this entry will override the global default set by the SWLDPRM entry.
3. The parameter CHKRUN can only be set on SWLDPRM and will be ignored on CONCTL entries.
4. A user fatal will be issued if DES on the SET3 entry is not "ELEM".

**CONM1**

## Concentrated Mass Element Connection, General Form

Defines a 6 x 6 symmetric mass matrix at a geometric grid point.

**Format:**

| 1     | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CONM1 | EID | G   | CID | M11 | M21 | M22 | M31 | M32 |    |
|       | M33 | M41 | M42 | M43 | M44 | M51 | M52 | M53 |    |
|       | M54 | M55 | M61 | M62 | M63 | M64 | M65 | M66 |    |

**Example:**

|       |     |      |   |     |     |  |  |  |      |
|-------|-----|------|---|-----|-----|--|--|--|------|
| CONM1 | 2   | 22   | 2 | 2.9 | 6.3 |  |  |  |      |
|       | 4.8 | 28.6 |   |     |     |  |  |  |      |
|       |     | 28.6 |   |     |     |  |  |  | 28.6 |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ ). See Remark 2. |
| G          | Grid point identification number. ( $\text{Integer} > 0$ ). See Remark 3.                   |
| CID        | Coordinate system identification number for the mass matrix. ( $\text{Integer} \geq 0$ )    |
| Mij        | Mass matrix values. (Real)  |

**Remarks:**

- For a less general means of defining concentrated mass at grid points, see the CONM2 entry description.
- Element identification numbers should be unique with respect to all other element identification numbers.
- Unlike the CONM2 entry, the CONM1 entry does not allow for the specification of concentrated mass at harmonic structural grid points. Therefore, grid point G must necessarily be a non-harmonic structural grid point.

**CONM2**

## Concentrated Mass Element Connection, Rigid Body Form

Defines a concentrated mass at a grid point.

Format:

| 1     | 2   | 3   | 4   | 5   | 6   | 7   | 8  | 9 | 10 |
|-------|-----|-----|-----|-----|-----|-----|----|---|----|
| CONM2 | EID | G   | CID | M   | X1  | X2  | X3 |   |    |
|       | I11 | I21 | I22 | I31 | I32 | I33 |    |   |    |

Example:

|       |      |    |      |      |  |     |  |  |  |
|-------|------|----|------|------|--|-----|--|--|--|
| CONM2 | 2    | 15 | 6    | 49.7 |  |     |  |  |  |
|       | 16.2 |    | 16.2 |      |  | 7.8 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ ). See Remark 1.  |
| G          | Grid point identification number. ( $\text{Integer} > 0$ ). See Remark 7.   |
| CID        | Coordinate system identification number. For CID of -1; see X1, X2, X3 below. ( $\text{Integer} \geq -1$ ; Default = 0). See Remarks 4., 6. and 7.  |
| M          | Mass value. (Real). See Remarks 5. and 7.   |
| X1, X2, X3 | Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real). See Remarks 4., 5., 6. and 7. |
| Iij        | Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (Real). See Remarks 4., 5. and 7.  |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.
3. The continuation is optional.
4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.

5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & \\ M & M & & \text{symmetric} \\ M & & M & \\ & I_{11} & & \\ & -I_{21} & I_{22} & \\ & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

where

$$\begin{aligned} M &= \int \rho dV \\ I_{11} &= \int \rho(x_2^2 + x_3^2) dV \\ I_{22} &= \int \rho(x_1^2 + x_3^2) dV \\ I_{33} &= \int \rho(x_1^2 + x_2^2) dV \\ I_{21} &= \int \rho x_1 x_2 dV \\ I_{31} &= \int \rho x_1 x_3 dV \\ I_{32} &= \int \rho x_2 x_3 dV \end{aligned}$$

and  $x_1, x_2, x_3$  are components of distance from the center of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is nonpositive definite, since this may cause fatal errors in dynamic analysis modules.

6. If CID  $\geq 0$ , then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.
7. MSC Nastran contains two kinds of structural grid points, namely, non-harmonic structural grid points and harmonic structural grid points. The former have the standard three translational and three rotational degrees of freedom while the latter have three symmetric components and three anti-symmetric components dependent on the harmonic value.

Harmonic grid points are identified automatically as the grid points listed on CQUADX and CTRIAX element entries that are associated with PAXSYMH entries.

For a harmonic grid point, the mass value M is the total mass. This value is not to be multiplied by  $2\pi$ . The mass matrix computed for harmonic grids is dependent on the harmonic value associated with the PAXSYMH entry and will be automatically determined by the code. Only harmonic values of 0 and 1 have contributions to grid point weight generator type calculations.

Any values specified in the CID, X1, X2, X3, I11, I21, I22, I31, I32, or I33 fields are ignored for a harmonic grid.

**CONROD**

## Rod Element Property and Connection

Defines a rod element without reference to a property entry.

**Format:**

| 1      | 2   | 3  | 4  | 5   | 6 | 7 | 8 | 9   | 10 |
|--------|-----|----|----|-----|---|---|---|-----|----|
| CONROD | EID | G1 | G2 | MID | A | J | C | NSM |    |

**Example:**

|        |   |    |    |   |      |  |  |  |  |
|--------|---|----|----|---|------|--|--|--|--|
| CONROD | 2 | 16 | 17 | 4 | 2.69 |  |  |  |  |
|--------|---|----|----|---|------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique element identification number. (0 < Integer < 100,000,000)               |
| G1, G2     | Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ) |
| MID        | Material identification number. (Integer > 0)                                   |
| A          | Area of the rod. (Real)   |
| J          | Torsional constant. (Real)  |
| C          | Coefficient for torsional stress determination. (Real)                          |
| NSM        | Nonstructural mass per unit length. (Real)                                      |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
4. For RC network solver in thermal analysis, the J, C and NSM are ignored.

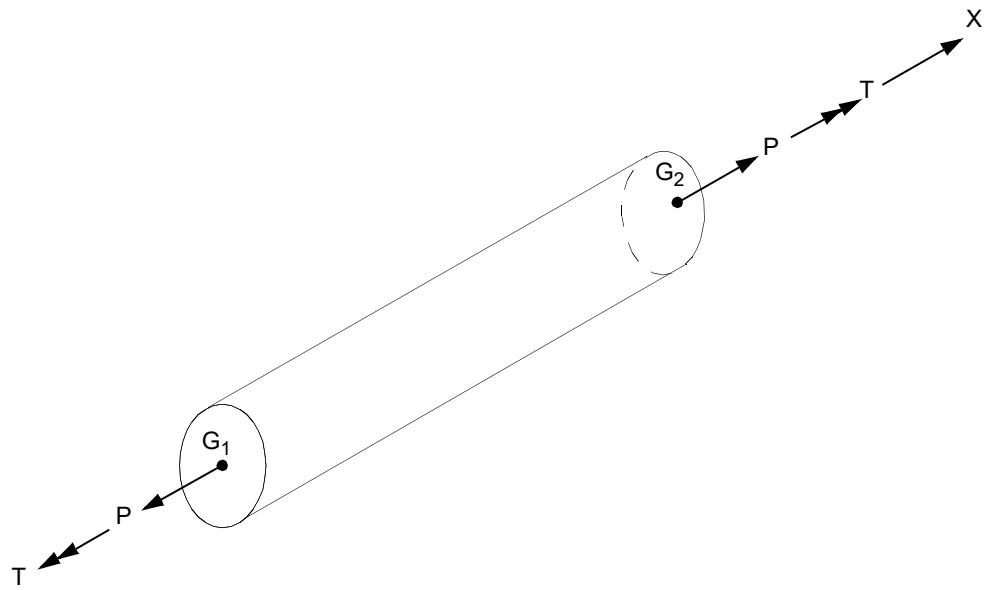


Figure 9-39 CONROD Element Forces and Moments

**CONTRLT**

## Thermal Control Element for Heat Transfer Analysis

Defines the control mechanism for QVECT, QVOL, QBDY3, RADBC, CONV, and CONVM in heat transfer analysis (SOL 159 and SOL 400 with analysis=htran).

Format:

| 1       | 2     | 3      | 4     | 5     | 6  | 7  | 8      | 9      | 10 |
|---------|-------|--------|-------|-------|----|----|--------|--------|----|
| CONTRLT | ID    | SENSOR | SFORM | CTYPE | PL | PH | PTYPE  | PZERO  |    |
|         | DT    | DELAY  | TAUC  |       |    |    | FCTMIN | FCTMAX |    |
|         | GAIN1 | GAIN2  | GAIN3 |       |    |    |        |        |    |

Example:

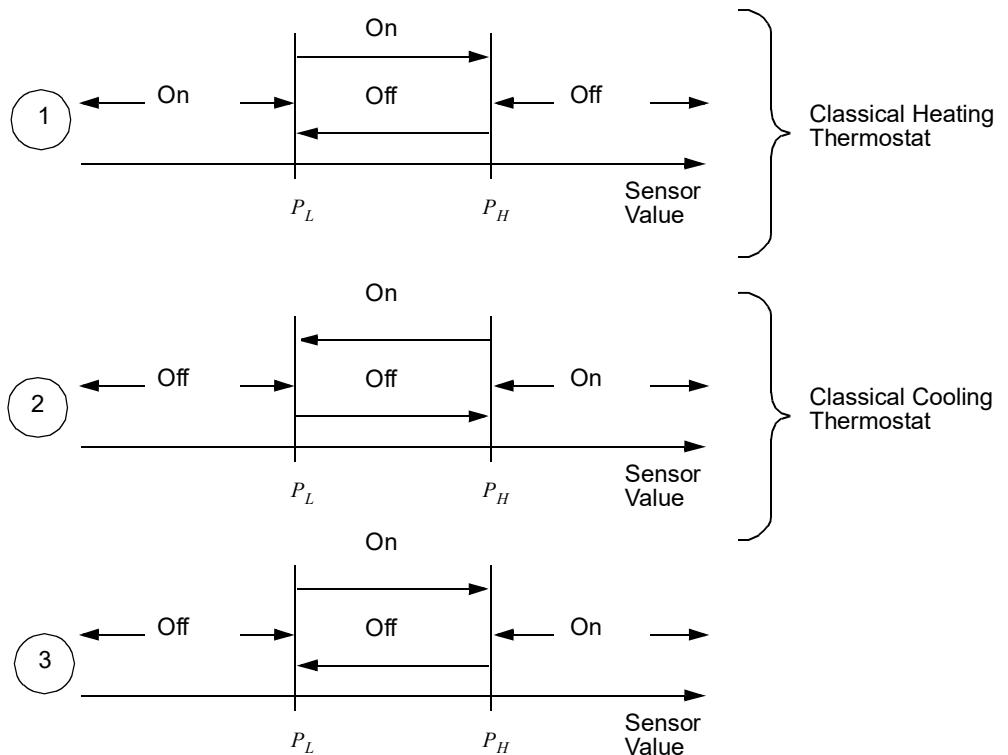
|         |     |      |    |       |     |     |    |    |  |
|---------|-----|------|----|-------|-----|-----|----|----|--|
| CONTRLT | 100 | 20   | T  | TSTAT | 68. | 73. | 7  | 1. |  |
|         | 0.1 | 0.01 | 0. |       |     |     | 0. | 1. |  |
|         | 0.  | 0.   | 0. |       |     |     |    |    |  |

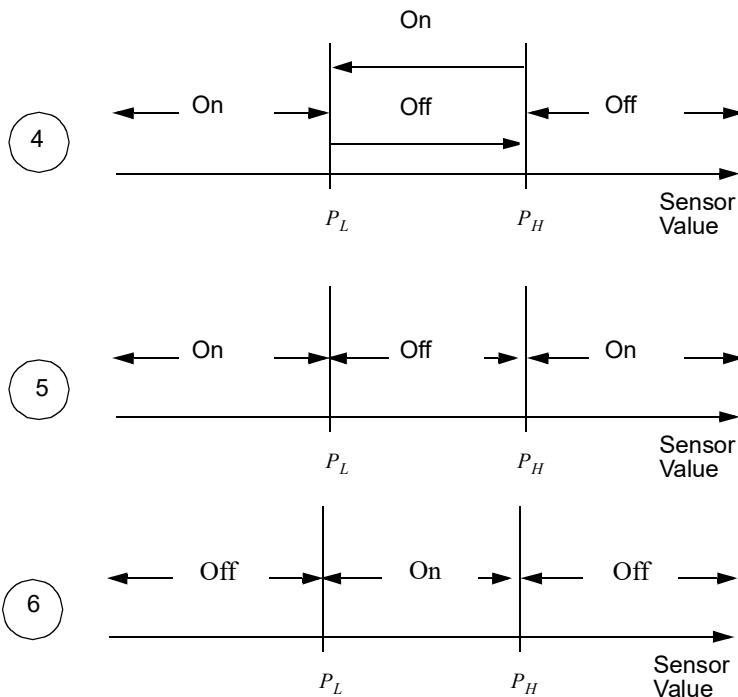
| Descriptor | Meaning  |  |
|------------|--|--|
| ID         | Controller identification node. See Remark 1. (Integer > 0; Required)                        |  |
| SENSOR     | Grid or scalar point ID of sensor. See Remark 2. (Integer > 0; Required)                     |  |
| SFORM      | Sensor output form. See Remark 3. (Character, T; Default = T)                                |  |
| CTYPE      | Control type. See Remark 4. (Character, Default TSTAT)                                       |  |
| PL         | Minimum temperature. For PTYPE 9 through 11, set temperature. See Remark 5. (Real; Required) |  |
| PH         | Maximum temperature. For PTYPE 9 through 11, unused. See Remark 5. (Real; Required)          |  |
| PTYPE      | Process type. See Remark 5. (Integer 1 through 11, 1 through 6 already defined; no Default)  |  |
|            | 7  | Bang-bang (a thermostat that is either on or off, a special case of a bang-bang controller)                    |
|            | 8  | Proportional   |
|            | 9  | Steady-state (special routine for steady state models)   |
|            | 10   | Classical PID (proportional-integral-differential) and does not include the time derivative of the set point ) |
|            | 11   | Standard PID (Includes the time derivative of the set point)   |
| PZERO      | Initial controller value. See Remark 4. (0. < Real < 1.; Default = 0)                        |  |
| DT         | Monitoring time interval or sampling period. (Real > 0.; Default = 0)                        |  |

| Descriptor | Meaning  |
|------------|--|
| DELAY      | Time delay after the switch is triggered or time for delayed control action in PID control. (Real > 0.; Default = 0.0)     |
| TAUC       | Decay time constant for actuator response. (Real > 0.; Default = 0.0)  |
| FCTMIN/MAX | Minimum factor/Maximum factor on load to apply. (Real; Default = 0.0)  |
| GAINx      | Gain variables for heat controller. Use GAIN1 for P value, GAIN2 for I value, and GAIN3 for D value. (Real; Default = 0.0) |

**Remarks:**

1. The CONTRLT ID is referenced by CNTRLND entry identified on any of the CONV, CONVM, RADBC, QVECT, QVOL, and QBDY3 Bulk Data entries. If any grid or scalar point ID is the same as the CONTRLT ID, then the combined logic associated with the controller and the control node will be in force for the LBC referenced. Any number of CONTRLT statements may exist in a single model.
2. Sensor point, where a feedback temperature or rate of change of temperature is measured. May be a dependent DOF in a MPC relationship.
3. Sensor output may only be temperature (T)
4. Control type can only be TSTAT. The PZERO field cannot have any other value but 0.0 or 1.0.
5. The upper and lower limit values (PL and PH) define a dead band for a thermostat. The available thermostat controller (TSTAT) formats are (PTYPE = 1 through 6).
6. RC network solver only supports PTTYPE=1 and PTTYPE=2 thermostat controller, SFORM, CTYPE, PZERO, DT, DELAY, TAUC and TA8 are ignored.





7. The last 5 parameters are for RC Network solver only.
8. RC Network solver only supports controllers (thermostats) on source data. This entry can be used in either steady-state (static) or transient state (dynamic). For more details about these parameters, please reference the *MSC SINDA User's Guide and Library Reference*.

**CONV****Heat Boundary Element Free Convection Entry**

Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

| 1    | 2   | 3      | 4     | 5       | 6   | 7   | 8   | 9   | 10 |
|------|-----|--------|-------|---------|-----|-----|-----|-----|----|
| CONV | EID | PCONID | FLMND | CNTRLND | TA1 | TA2 | TA3 | TA4 |    |
|      | TA5 | TA6    | TA7   | TA8     |     |     |     |     |    |

Example:

|      |   |     |   |     |     |  |  |  |  |
|------|---|-----|---|-----|-----|--|--|--|--|
| CONV | 2 | 101 | 3 | 201 | 301 |  |  |  |  |
|------|---|-----|---|-----|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | CHBDYG, CHBDYE, or CHBDYP surface element identification number.<br>(0 < Integer < 100,000,000)                                    |
| PCONID     | Convection property identification number of a PCONV entry. (Integer > 0)  |
| FLMND      | Point for film convection fluid property temperature. (Integer ≥ 0; Default = 0)   |
| CNTRLND    | Control point for free convection boundary condition. (Integer ≥ 0; Default = 0)   |
| TAi        | Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2 through TA8; Default for TA2 through TA8 is TA1.) |

Remarks:

- The basic exchange relationship can be expressed in one of the following forms:
  - $q = H \cdot (T - TAMB)^{EXPF} (T - TAMB)$ , CNTRLND = 0
  - $q = (H \cdot u_{CNTRLND})(T - TAMB)^{EXPF} (T - TAMB)$ , CNTRLND ≠ 0
  - $q = H(T^{EXPF} - TAMB^{EXPF})$ , CNTRLND = 0
  - $q = (H \cdot u_{CNTRLND})(T^{EXPF} - TAMB^{EXPF})$ , CNTRLND ≠ 0
 EXPF is specified on the PCONV entry.  
(See [PCONV, 2801](#) entry for additional clarification of forms.)
- The continuation entry is not required.
- CONV is used with an CHBDYi (CHBDYG, CHBDYE, or CHBDYP) entry having the same EID.
- The temperature of the film convection point provides the look up temperature to determine the convection film coefficient. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperature, or the ambient temperature, as defined in the FORM field of the PCONV Bulk Data entry.

5. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If midside ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
6. See the Bulk Data entry, [PConv, 2801](#), for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.
7. RC network solver only supports CNTRLND node defined by SPC or SPCD entries, FLMND is ignored.

**CONVM****Heat Boundary Element Forced Convection Entry**

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

**Format:**

| 1     | 2   | 3      | 4     | 5       | 6   | 7   | 8    | 9 | 10 |
|-------|-----|--------|-------|---------|-----|-----|------|---|----|
| CONVM | EID | PCONID | FLMND | CNTMDOT | TA1 | TA2 | Mdot |   |    |

**Example:**

|       |     |   |     |     |    |    |  |  |  |
|-------|-----|---|-----|-----|----|----|--|--|--|
| CONVM | 101 | 1 | 201 | 301 | 20 | 21 |  |  |  |
|-------|-----|---|-----|-----|----|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | CHBDYP element identification number. (0 < Integer < 100,000,000)   |
| PCONID     | Convection property identification number of a PCONVM entry. (Integer > 0)  |
| FLMND      | Point used for fluid film temperature. (Integer $\geq 0$ ; Default = 0)   |
| CNTMDOT    | Control point used for controlling mass flow. (Integer $\geq 0$ or Blank). Blank or zero is only allowed when Mdot > 0.0. See Remark 3.                 |
| TA1, TA2   | Ambient points used for convection. (Integer > 0 for TA1 and Integer $\geq 0$ for TA2; Default for TA2 is TA1.)   |
| Mdot       | A multiplier for mass flow rate in case there is no point associated with the CNTRLND field. (Real > 0.0; Default = 1.0 if CNTMDOT > 0). See Remark 3.) |

**Remarks:**

1. CONVM is used with an CHBDYP entry of type FTUBE having the same EID.
2. The temperature of the fluid film point may be specified to determine the material properties for the fluid. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperatures, or the ambient temperature, as defined in the FORM field of the PCONVM Bulk Data entry.
3. The CNTMDOT has a dual function. It can reference the ID of the CONTRLT Bulk Data entry to activate a thermostat controlled flux transfer. It can also reference a GRID or a scalar point which is set to the desired mass flow rate (mdot) to effect the advection of energy downstream at an  $mdot \cdot Cp \cdot T$  rate. If CNTMDOT is zero or is not specified (blank field), then the mass flow rate must be specified on the Mdot field (8th field).

Case 1: Define a thermostat controller using the CONTRLT Bulk Data entry.

Total mass flow rate =  $\text{CONTRLT} \cdot \text{Mdot} \cdot \text{CNTMDOT}$

where       $\text{CONTRLT}$  is the grid ID on the 2nd field of the  $\text{CONTRLT}$  Bulk Data entry that ties the mass flow rate through this sensor grid.

$\text{Mdot}$  is the 8th field of the  $\text{CONVM}$  Bulk Data entry.

$\text{CNTMDOT}$  is the grid ID that has a mass flow rate associated with it.

Example: The thermostat controller controls GRID 5 (sensor grid) from 250 degrees to 300 degrees. If the temperature exceeds 300 degrees, then the mass flow is turned ON. If the temperature is below 250 degrees, then the mass flow is turned OFF.

```
CONTRLT, 5, 2, T, TSTAT, 250.0, 300.0, 1, 1.0 +
+, 0.0, 0.0, 0.0
CONVM, 100001, 1, , 5, 4
```

Note that GRID 5 is also specified on the 5th field of the  $\text{CONVM}$  Bulk Data entry.

Case 2: There is no  $\text{CONTRLT}$  Bulk Data entry (more general case).

Total mass flow rate =  $\text{Mdot} \cdot \text{CNTMDOT}$

If the  $\text{CNTMDOT}$  (the 5th field of the  $\text{CONVM}$  Bulk Data entry) is specified, then the default  $\text{Mdot}$  value is 1.0 and the mass flow rate is defined through  $\text{CNTMDOT}$ .

Example: Define a constant mass flow rate of 0.5 through  $\text{CNTMDOT}$ .

```
CONVM, 100001, 1, , 5, 4
SPC, 1, 5, 1, 0.5
```

To specify a time-varying mass flow rate, replace the  $\text{SPC}$  Bulk Data entry by  $\text{TEMPBC,TRAN}$  for SOL 159 or by  $\text{SPC1}$  and  $\text{SPCD}$  for SOL 400.

For SOL 159:

```
TEMPBC, 2, TRAN, 1.0, 5
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , , 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

For SOL 400:

```
SPC = 111 (Case Control command)
SPC1, 111, , 5
SPCD, 2, 5, , 1.0
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , 1, 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

If the  $\text{CNTMDOT}$  is not specified (zero or blank), then the mass flow rate must be constant and must be defined on  $\text{Mdot}$  (the 8th field of the  $\text{CONVM}$  Bulk Data entry).

Example: Define a mass flow rate of 0.5 directly on  $\text{Mdot}$  without specifying  $\text{CNTMDOT}$ .

CONVM, 100001, 1,,, 99,, 0.5

4. If only the first ambient point is specified then, the second ambient point is assumed to have the same temperature.
5. See the Bulk Data entry, [PCONVM, 2804](#), for an explanation of the mathematical relationships available for forced convection and the reference temperature for fluid material properties.
6. RC network solver does not support CONVM for thermal analysis, it uses PRJCON for forced convection.

**CORD1C**

## Cylindrical Coordinate System Definition, Form 1

Defines a cylindrical coordinate system using three grid points.

**Format:**

| 1      | 2    | 3   | 4   | 5   | 6    | 7   | 8   | 9   | 10 |
|--------|------|-----|-----|-----|------|-----|-----|-----|----|
| CORD1C | CIDA | G1A | G2A | G3A | CIDB | G1B | G2B | G3B |    |

**Example:**

|        |   |    |    |    |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|
| CORD1C | 3 | 16 | 32 | 19 |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CIDA, CIDB | Coordinate system identification number. (Integer > 0)                               |
| GiA, GiB   | Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A ; G1B ≠ G2B ≠ G3B ) |

**Remarks:**

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in Figure 9-40) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.

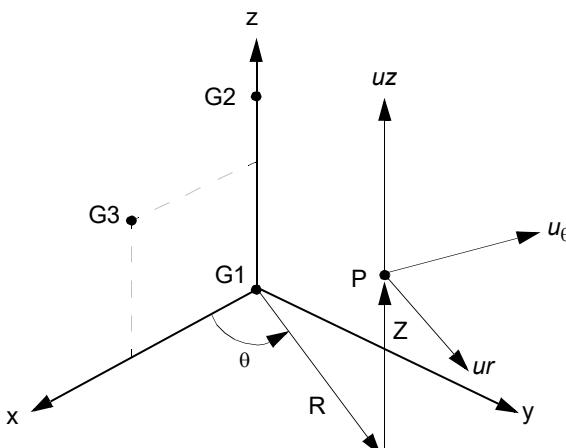


Figure 9-40      CORD1C Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_z)$ .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.

If CYLINDRICAL and point exactly on z-axis ( $r = 0.0$  exactly) then R,  $\theta$ , Z triad coordinates are exactly aligned with the local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

**CORD1R**

## Rectangular Coordinate System Definition, Form 1

Defines a rectangular coordinate system using three grid points.

**Format:**

| 1      | 2    | 3   | 4   | 5   | 6    | 7   | 8   | 9   | 10 |
|--------|------|-----|-----|-----|------|-----|-----|-----|----|
| CORD1R | CIDA | G1A | G2A | G3A | CIDB | G1B | G2B | G3B |    |

**Example:**

|        |   |    |    |    |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|
| CORD1R | 3 | 16 | 32 | 19 |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CIDA, CIDB | Coordinate system identification number. (Integer > 0)                                 |
| GiA, GiB   | Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B ) |

**Remarks:**

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in Figure 9-41) in this coordinate system is given by (X, Y, Z).

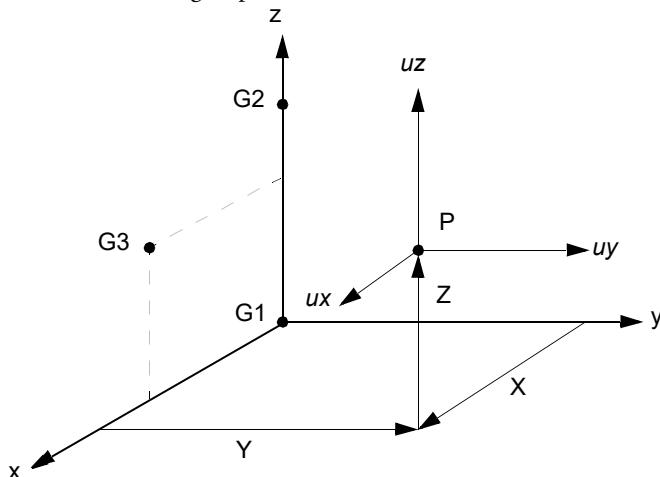


Figure 9-41 CORD1R Definition

5. The displacement coordinate directions at P are shown above by  $(u_x, u_y, u_z)$ .

**CORD1S**

## Spherical Coordinate System Definition, Form 1

Defines a spherical coordinate system by reference to three grid points.

Format:

| 1      | 2    | 3   | 4   | 5   | 6    | 7   | 8   | 9   | 10 |
|--------|------|-----|-----|-----|------|-----|-----|-----|----|
| CORD1S | CIDA | G1A | G2A | G3A | CIDB | G1B | G2B | G3B |    |

Example:

|        |   |    |    |    |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|
| CORD1S | 3 | 16 | 32 | 19 |  |  |  |  |  |
|--------|---|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CIDA, CIDB | Coordinate system identification numbers. (Integer > 0)                                |
| GiA, GiB   | Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B ) |

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with a definition that does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in Figure 9-42) in this coordinate system is given by  $(R, \theta, \phi)$  where  $\theta$  and  $\phi$  are measured in degrees.

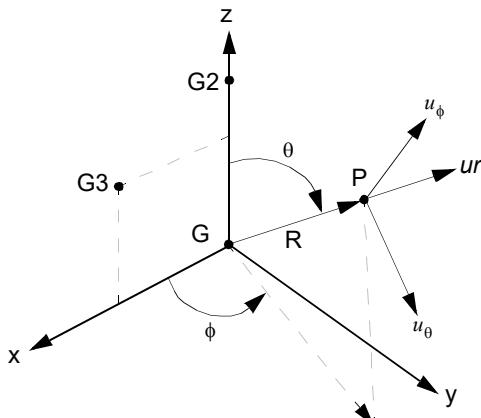


Figure 9-42 CORD1S Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_\phi)$ .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.

If SPHERICAL and R = 0.0 (point at origin) then R,  $\theta$ ,  $\phi$  triad coordinates are exactly aligned with the local defining system triad. User caution: if  $R \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

If SPHERICAL and r in azimuth plane is  $r = 0.0$  and  $z \neq 0.0$  then the coordinate system is defined as R global along  $\pm Z$ - local defining system triad according if  $z > 0.$  or  $z < 0.$ ,  $\theta$  -global along X-local defining system triad and  $\phi$  -global along Y-local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

**CORD2C****Cylindrical Coordinate System Definition, Form 2**

Defines a cylindrical coordinate system using the coordinates of three points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|----|----|----|----|----|----|
| CORD2C | CID | RID | A1 | A2 | A3 | B1 | B2 | B3 |    |
|        | C1  | C2  | C3 |    |    |    |    |    |    |

**Example:**

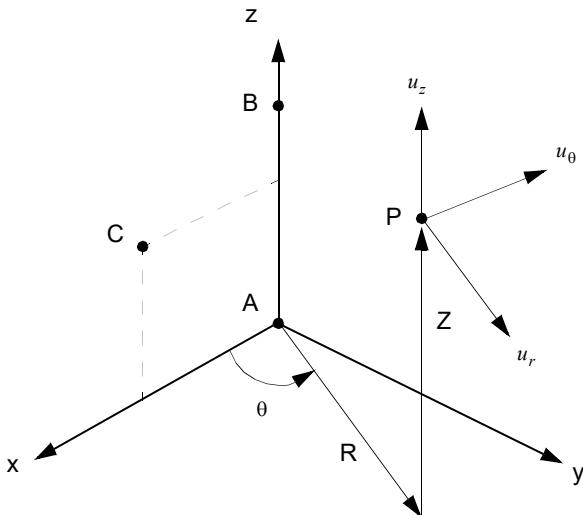
|        |     |     |      |     |     |     |     |     |  |
|--------|-----|-----|------|-----|-----|-----|-----|-----|--|
| CORD2C | 3   | 17  | -2.9 | 1.0 | 0.0 | 3.6 | 0.0 | 1.0 |  |
|        | 5.2 | 1.0 | -2.9 |     |     |     |     |     |  |

| Descriptor | Meaning  |
|------------|--|
| CID        | Coordinate system identification number. (Integer > 0)   |
| RID        | Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq 0$ ; Default = 0 is the basic coordinate system.) |
| Ai, Bi, Ci | Coordinates of three points given with respect to the coordinate system defined by RID. (Real)   |

**Remarks:**

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in [Figure 9-43](#)) in this coordinate system is given by ( $R, \theta, Z$ ), where  $\theta$  is measured in degrees.



**Figure 9-43** CORD2C Definition

6. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_z)$ .
7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Manual*. If CYLINDRICAL and point exactly on z-axis ( $r = 0.0$  exactly) then  $R, \theta, Z$  triad coordinates are exactly aligned with the local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.
9. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

**CORD2R****Rectangular Coordinate System Definition, Form 2**

Defines a rectangular coordinate system using the coordinates of three points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|----|----|----|----|----|----|
| CORD2R | CID | RID | A1 | A2 | A3 | B1 | B2 | B3 |    |
|        | C1  | C2  | C3 |    |    |    |    |    |    |

**Example:**

|        |     |     |      |     |     |     |     |     |  |
|--------|-----|-----|------|-----|-----|-----|-----|-----|--|
| CORD2R | 3   | 17  | -2.9 | 1.0 | 0.0 | 3.6 | 0.0 | 1.0 |  |
|        | 5.2 | 1.0 | -2.9 |     |     |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| CID        | Coordinate system identification number. (Integer > 0)  |
| RID        | Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq 0$ ; Default = 0; which is the basic coordinate system.) |
| Ai, Bi, Ci | Coordinates of three points given with respect to the coordinate system defined by RID. (Real)  |

**Remarks:**

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

6. The location of a grid point (P in the [Figure 9-44](#)) in this coordinate system is given by (X, Y, Z).

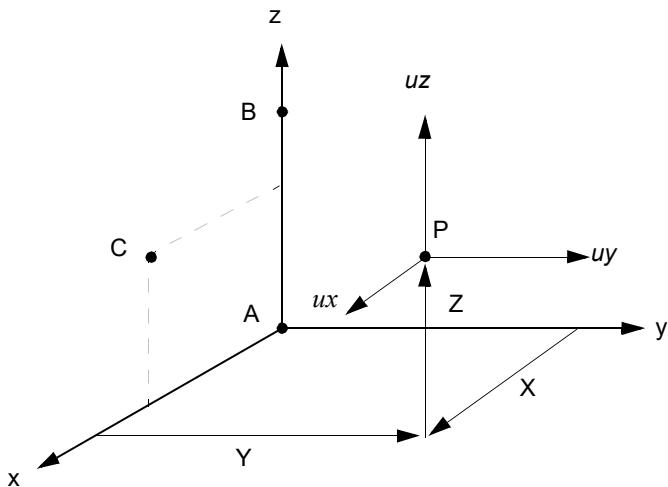


Figure 9-44 CORD2R Definition

7. The displacement coordinate directions at P are shown by  $(u_x, u_y, u_z)$ .
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

**CORD2S****Spherical Coordinate System Definition, Form 2**

Defines a spherical coordinate system using the coordinates of three points.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|----|----|----|----|----|----|
| CORD2S | CID | RID | A1 | A2 | A3 | B1 | B2 | B3 |    |
|        | C1  | C2  | C3 |    |    |    |    |    |    |

**Example:**

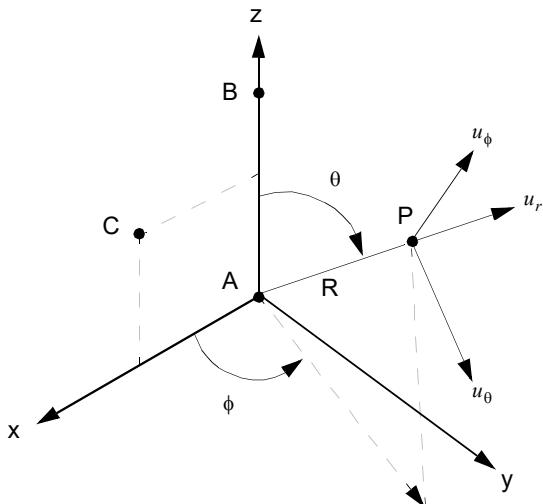
|        |     |     |      |     |     |     |     |     |  |
|--------|-----|-----|------|-----|-----|-----|-----|-----|--|
| CORD2S | 3   | 17  | -2.9 | 1.0 | 0.0 | 3.6 | 0.0 | 1.0 |  |
|        | 5.2 | 1.0 | -2.9 |     |     |     |     |     |  |

| Descriptor | Meaning  |
|------------|--|
| CID        | Coordinate system identification number. (Integer > 0)   |
| RID        | Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq 0$ ; Default = 0 is the basic coordinate system.) |
| Ai, Bi, Ci | Coordinates of three points given with respect to the coordinate system defined by RID. (Real)   |

**Remarks:**

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must all be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

6. The location of a grid point (P in [Figure 9-45](#)) in this coordinate system is given by  $(R, \theta, \phi)$ , where  $\theta$  and  $\phi$  are measured in degrees.



**Figure 9-45** CORD2S Definition

7. The displacement coordinate directions at P are shown above by  $(u_r, u_\theta, u_\phi)$ .
8. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.
- If SPHERICAL and  $R = 0.0$  (point at origin) then  $R, \theta, \phi$  triad coordinates are exactly aligned with the local defining system triad. User caution: if  $R \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.
- If SPHERICAL and r in azimuth plane is  $r = 0.0$  and  $z \neq 0.0$  then the coordinate system is defined as R global along  $\pm Z$ -local defining system triad according if  $z > 0.$  or  $z < 0.$ ,  $\theta$ -global along X-local defining system triad and  $\phi$ -global along Y-local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.
9. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

**CORD3G****General Coordinate System**

Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. The CORD3G entry is used with the MAT9 entry to orient material principal axes for 3-D composite analysis.

Format:

| 1      | 2   | 3      | 4    | 5        | 6        | 7        | 8      | 9 | 10 |
|--------|-----|--------|------|----------|----------|----------|--------|---|----|
| CORD3G | CID | METHOD | FORM | THETAID1 | THETAID2 | THETAID3 | CIDREF |   |    |

Example:

|        |     |      |     |     |     |     |   |  |  |
|--------|-----|------|-----|-----|-----|-----|---|--|--|
| CORD3G | 100 | E313 | EQN | 110 | 111 | 112 | 0 |  |  |
|--------|-----|------|-----|-----|-----|-----|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| CID        | Coordinate system identification number. See Remark 1. (Integer > 0)   |
| METHOD     | E313 or S321 for Euler angle rotation in 313 sequence or space-fixed rotation in 321 sequence. See Remark 2. (Character; Default = "E313")   |
| FORM       | Specifies the Bulk Data entry which defines angles. FORM = "EQN" for DEQATN entry or FORM = "TABLE" for TABLE3D entry. (Character; Default = "EQN")  |
| THETAID    | Identification number for DEQATN or TABLE3D Bulk Data entry which defines the three angles (in radians) measured from reference coordinates to the general material coordinate system. See Remark 3. (Integer > 0) |
| CIDREF     | Identification number for the coordinate system from which the orientation of the general coordinate system is defined. (Integer $\geq 0$ ; Default = 0)   |

Remarks:

1. CID must be unique with respect to all other coordinate systems. CID cannot be referenced on GRID entries.
2. Three Euler angles specify the rotation of the CORD3G coordinate axes (xyz) with respect to the local Cartesian coordinate axes (XYZ) in CIDREF as follows: first rotate about Z-axis by  $\theta_1$ , next rotate about rotated x-axis by  $\theta_2$ , and then rotate about rotated z-axis by  $\theta_3$ . On the other hand, the space-fixed rotations in 321 sequence specify all the rotations about the fixed coordinate axes: first rotate about Z by  $\theta_1$ , next about Y by  $\theta_2$ , then about X by  $\theta_3$ .
3. The three rotations define a coordinate transformation which transforms position vectors in the reference coordinate system into the general coordinate system.
4. The DEQATN option must have three arguments representing the three axes of CIDREF, although not all arguments are necessarily needed in the equation.

5. If FORM = EQN is used with CIDREF pointing to any CORD1k or CORD2k, k = C,S, and a spatial relationship is required, then the equations are written using  $R, \theta, \varphi$  as input variables. Non spatial fixed values such as THETA1 (A,B,C)=PI(1.) may of course be entered for any type of coordinate defined by the CIDREF.

If FORM = TABLE3D is used with CIDREF pointing to any CORD1k or CORD2k, k = C,S, and a spatial relationship is required then the equations are written using the variables X,Y, Z defined at the origin of the CIDREF as input values and not the variables  $R, \theta, Z$  or  $R, \theta, \varphi$  input values. For example, if a CORD1S is used then the table input value is X internally computed as  $R \cdot \sin\theta \cdot \cos\varphi$ .

Whether the EQN or TABLE3D FORM is used, the equation must return an angle measure in radians to have any meaning.

**CORD3R****Moving Rectangular Coordinate System**

Defines a moving rectangular coordinate system using three points in SOL 700 only.

**Format:**

| 1      | 2   | 3  | 4  | 5  | 6   | 7  | 8  | 9  | 10 |
|--------|-----|----|----|----|-----|----|----|----|----|
| CORD3R | CID | N1 | N2 | N3 | CID | N1 | N2 | N3 |    |

**Example:**

|        |      |   |     |     |  |  |  |  |  |
|--------|------|---|-----|-----|--|--|--|--|--|
| CORD3R | 1001 | 1 | 144 | 300 |  |  |  |  |  |
|--------|------|---|-----|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CID        | Unique coordinate system number. (Integer > 0)     |
| N1, N2, N3 | Grid point numbers (must be unique). (Integer > 0) |

**Remarks:**

1. Available in SOL 700 only.
2. Two different coordinate systems may be defined on one entry.
3. The grid points must be defined in an independent coordinate system.
4. The first grid point is the origin, the second lies on the z-axis and the third lies in the x-z plane.
5. The position and orientation of the coordinate system is updated as the grid points move.
6. The three grid points must not be colinear.

**COUCOHF****Cohesive Friction Model To Be Used For a COUPLE Entry**

Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface. Use SOL700 only.

**Format:**

|         |     |         |       |           |  |  |  |  |
|---------|-----|---------|-------|-----------|--|--|--|--|
| COUCOHF | CID | COHFRID | SUBID | COHFRICID |  |  |  |  |
|---------|-----|---------|-------|-----------|--|--|--|--|

**Example:**

|         |     |    |    |    |  |  |  |  |
|---------|-----|----|----|----|--|--|--|--|
| COUCOHF | 112 | 14 | 12 | 12 |  |  |  |  |
|---------|-----|----|----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| CID        | Unique number of a COUCOHF entry. (Integer $\geq 0$ ; required)   |
| COHFRID    | Number of a set of COUCOHF entries. COUCOHF must be referenced from a COUPLE entry. (Integer $\geq 0$ ; required)                       |
| SUBID      | BSURF, BCSEG or BCPROP id, which must be a part of the BSID field of the COUPLE entry. See Remark 2. (Integer $\geq 0$ ; default=blank) |
| COHFRICID  | Cohesive friction ID (Integer $\geq 0$ ; required)  |

**Remarks:**

1. One couple entry can reference more than one COUCOHF entry. This allows a cohesive friction that varies along the coupling surface.
2. When SUBID is left blank, then the cohesive friction applies to the whole coupling surface.
3. A coupling surface segment can only have one porosity or one cohesive friction definition assigned.

## COUOPT

## Coupling Surface Pressure Definition

Defines the interaction factor and a pressure load from the covered side acting on a BSURF, BCPROP, BCMATL, BCSEG, BCBOX. Used in SOL 700 only.

Format:

| 1      | 2       | 3        | 4     | 5      | 6       | 7 | 8 | 9 | 10 |
|--------|---------|----------|-------|--------|---------|---|---|---|----|
| COUOPT | CID     | OPTID    | SUBID | FACTOR | FACTORY |   |   |   |    |
|        | PLCOVER | PLCOVERV |       |        |         |   |   |   |    |

Example:

|        |          |      |    |          |  |  |  |  |  |
|--------|----------|------|----|----------|--|--|--|--|--|
| COUOPT | 1        | 80   | 42 | CONSTANT |  |  |  |  |  |
|        | CONSTANT | 1.E5 |    |          |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| CID        | Unique number of a COUOPT entry. (Integer > 0; Required)   |
| OPTID      | Number of a set of COUOPT entries. OPTID must be referenced from a COUPLE entry. (Integer > 0; Required)   |
| SUBID      | <p>&gt; 0      Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be part of the surface as defined in the COUPLE entry. (Integer <math>\geq 0, 0</math>)</p> <p>= 0      COUOPT definitions used for the entire surface as defined in the COUPLE entry.</p>              |
| FACTOR     | <p>Method of defining the interaction FACTORV with which the Eulerian pressure acting on the surface is multiplied. (Character, CONSTANT)</p> <p>CONSTANT   The FACTOR is constant and specified in FACTORV</p>  |
| FACTORV    | The interaction factor. (Real, 1.)   |
| PLCOVER    | <p>Method of defining the pressure load exerted on the faces of the surface from the covered side. The pressure load is applied only when the Eulerian pressure is greater than zero. (Character, CONSTANT)</p> <p>CONSTANT   The PLCOVER is constant and specified in PLCOVERV.</p> |
| TABLE      | The PLCOVER varies with time. PLCOVERV is the number of a TABLED1 entry giving the variation of the PLCOVER (y-value) with time (x-value).   |
| PLCOVERV   | The pressure load or the number of a TABLED1 entry depending on the PLCOVER entry. (Real $\geq 0, 0$ .)  |

**Remarks:**

1. The effect of specifying an interaction FACTOR is similar to specifying a porosity coefficient on a COUPOR entry. The difference is that in this case the surface still acts as a wall boundary for the Eulerian material.
2. Applying a PLCOVER instead of applying a pressure load on the faces through either a PLOAD, PLOAD4, or DAREA entry gives the following differences:
  - a. PLCOVER is applied only when there is a balancing Eulerian pressure greater than zero.
  - b. Possible porosity as defined on a COUPOR entry is taken into account when applying the PLCOVER.
  - c. With PARAM,PLCOVCUT you can define a cut-off time that is applied to PLCOVER.
3. The covered side of a surface lies on the side where there is no Eulerian material.

**COUP1FL****Coupling Surface Failure**

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

**Format:**

| 1       | 2    | 3   | 4   | 5    | 6    | 7    | 8        | 9        | 10 |
|---------|------|-----|-----|------|------|------|----------|----------|----|
| COUP1FL | CFID | RHO | SIE | XVEL | YVEL | ZVEL | PRESSURE | MATERIAL |    |

**Example:**

|         |   |       |         |      |  |  |  |  |  |
|---------|---|-------|---------|------|--|--|--|--|--|
| COUP1FL | 3 | 1.225 | 204082. | 900. |  |  |  |  |  |
|---------|---|-------|---------|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| CFID       | Unique ID of a COUP1FL entry referenced from the COUPLE entry. (Integer > 0; Required) |
| RHO        | Surrounding density. See Remark 2. (Real > 0)  |
| SIE        | Surrounding specific internal energy. See Remark 2. (Real)                             |
| XVEL       | Surrounding x-velocity. See Remark 2. (Real)   |
| YVEL       | Surrounding y-velocity. See Remark 2. (Real)   |
| ZVEL       | Surrounding z-velocity. See Remark 2. (Real)   |
| PRESSURE   | Surrounding pressure. See Remark 4. (Real)   |
| MATERIAL   | MATDEUL ID. Only used when the multi-material Euler solver is active. (Blank)          |

**Remarks:**

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. For restrictions on the use of COUP1FL refer to param,flow-method. The coupling surface failure is associated with the element failure of the structure to which the surface is connected. Therefore, you have to define a failure model for the structure for the entry to take effect in the analysis.
2. For the first order Euler solvers no restriction apply to the values of the surrounding variables. For the Roe solver at least one of the surrounding variables should be defined. The default value of the density (RHO) will be set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
3. The coupling surface must only consist of CQUAD and/or CTRIA elements.
4. The field PRESSURE has to be left blank in combination with the Roe solver.
5. In combination with multi-material Euler only outflow of material is allowed. Each material in an outflow Euler element is transported. The materials are transported in proportion to their relative volume fractions.

**COUPINT**

## Coupling Surface Interaction

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3    | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|------|------|---|---|---|---|---|----|
| COUPINT | CID | CID1 | CID2 |   |   |   |   |   |    |

**Example:**

|         |    |   |  |  |  |  |  |  |  |
|---------|----|---|--|--|--|--|--|--|--|
| COUPINT | 33 | 2 |  |  |  |  |  |  |  |
|---------|----|---|--|--|--|--|--|--|--|

| Descriptor | Meaning                           |
|------------|-----------------------------------|
| CIID       | Unique number of a COUPINT entry. |
| CID1       | Number of COUPLE entry 1.         |
| CID2       | Number of COUPLE entry 2.         |

**Remarks:**

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. The interaction will be activated when failure of a Lagrangian structure with which the coupling surface is associated occurs. Therefore, you have to define a failure model for the material of the structure.
2. The coupling surface must consist of CQUAD and/or CTRIA elements.

**COUPLE****General Euler-Lagrange Coupling Surface**

Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain. Used in SOL 700 only.

Format:

| 1      | 2      | 3      | 4     | 5       | 6       | 7      | 8       | 9     | 10 |
|--------|--------|--------|-------|---------|---------|--------|---------|-------|----|
| COUPLE | CID    | BSID   | COVER | REVERSE | CHECK   | PORID  | OPTID   | CTYPE |    |
|        | INFID  | HTRID  | FS    | FK      | EXP     | INTID  |         |       |    |
|        | SET1ID | MESHID | TDEAC | COUP1FL | HYDSTAT | SKFRIC | COHFRID |       |    |

Example:

|        |     |    |        |     |     |  |  |  |  |
|--------|-----|----|--------|-----|-----|--|--|--|--|
| COUPLE | 100 | 37 | INSIDE | ON  | ON  |  |  |  |  |
|        |     |    |        | 0.3 | 0.0 |  |  |  |  |

| Descriptor | Meaning   |  |
|------------|---|--|
| CID        | Unique number of a COUPLE entry. (Integer > 0; Required)  |  |
| BSID       | Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG entry defining the coupling surface. (Integer > 0; Required)  |  |
| COVER      | The processing strategy for Eulerian elements inside and outside of the coupling surface.<br>(Character, INSIDE)  |  |
|            | INSIDE      The Eulerian elements inside the closed volume of the coupling surface are not processed.   |  |
|            | OUTSIDE     The Eulerian elements outside the closed volume of the coupling surface are not processed   |  |
|            | NONE        The Eulerian elements on both sides of the coupling surface are processed.  |  |
| REVERSE    | Auto-reverse switch for the coupling surface segments. (Character, ON)  |  |
|            | ON           If required, the normals of the coupling surface segments are automatically reversed so that they all point in the same general direction as to give a positive closed volume. |  |
|            | OFF          The segments are not automatically reversed. The user is responsible for defining the correct general direction of the segment normals.  |  |
| CHECK      | Check switch for coupling surface segments. (Character, ON)   |  |
|            | ON           The normals of the segments are checked to verify that they all point in the same general direction and yield a positive closed volume.  |  |
|            | OFF          The segments are not checked. It is the responsibility of the user to ensure that the direction of the segment normals yield a positive closed volume.                         |  |

| Descriptor | Meaning   |
|------------|---|
|            | When “REVERSE” is set to “ON”, the “CHECK” option will be automatically activated.  |
| PORID      | Number of a set of “LEAKAGE” entries that define the porosity for the BSURF entries. (Integer > 0, 0 (no porosity))   |
| OPTID      | Not available for the Roe solver<br>Number of a set of “COUOPT” entries that define special options for the BSURF entries (Integer $\geq 0$ , 0 (no special options))   |
| CTYPE      | Not available for the Roe solver<br>Coupling surface type definition. (Character, STANDARD)<br>STANDARD Standard Euler-Lagrange interaction.<br>AIRBAG Coupling for airbag applications.<br>It is equivalent to the standard coupling algorithm with the following exceptions that tailor the solution for airbag applications:<br>Inflow through a porous (sub)-surface will only occur when there is already some material (gas) in the Eulerian element.<br>Almost empty Eulerian elements will be automatically eliminated. The standard algorithm redistributes the small mass to the most suitable neighbor elements. |
| INFID      | Number of a set of “ABINFL” entries that define the inflator(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer > 0, 0 (no inflators))   |
| HTRID      | Number of a set of “HEATLOS” entries that define the heat transfer definition(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer $\geq 0$ , 0 (no heat transfer))  |
| FS         | Static friction coefficient. See Remark 6. Not available for the Roe solver. (Real $\geq 0.0$ , 0.0)  |
| FK         | Kinetic friction coefficient. See Remark 6. Not available for the Roe solver. (Real $\geq 0.0$ , 0.0)   |
| EXP        | Exponential decay coefficient. See Remark 6. Not available for the Roe solver. (Real $\geq 0.0$ , 0.0)  |
| INTID      | ID of an INITGAS entry specifying initial gas composition for the Euler mesh. See Remark 7. (Integer $\geq 0$ , 0 (no initial gas composition))   |
| SET1ID     | The number of a SET1 entry, which defines the Eulerian elements associated with this coupling surface. See Remark 8. (Integer $\geq 0$ ; Default = blank)   |
| MESHID     | ID of a MESH entry, which defines the Eulerian mesh associated with this coupling surface. See Remark 8. (Integer $\geq 0$ ; Default = blank)   |
| TDEAC      | Time of deactivation of the coupling surface and the associated Eulerian mesh. (Real $\geq 0.0$ , 1.E20)  |

| Descriptor | Meaning   |
|------------|---|
| COUP1FL    | The number of a COUP1FL entry, which defines the surrounding variables for the coupling surface when its segments fail. See Remark 9. (Integer $\geq 0$ ; Default = blank)  |
| HYDSTAT    | The number of a HYDSTAT entry, which specifies a hydrostatic preset. The preset is applied to all Euler element specified by the SET1ID and MESHID. See Remark 11. (Integer $\geq 0$ , 0 (no hydrostatic preset)) |
| SKFRIC     | Skin friction value. See Remark 13. (Real $\geq 0.0, 0.0$ )   |
| COHFRID    | Number of a set of COUCOHF entries that define the cohesive friction on the subsurface(s) of the coupling surface. (I $\geq 0$ Default = blank)   |

**Remarks:**

1. All coupling surfaces must form a multi-faceted closed volume. If necessary, additional segments must be defined to achieve the closed volume. The closed volume must intersect at least one Euler element initially.
2. All segments must be attached to the face of an element. Dummy elements can be used to define any additional segments that are required to create the closed volume.
3. The normals of all segments that form the coupling surface must point in the same general direction and result in a positive closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of the initial definition of the segments.
4. The “COVER” field determines how Eulerian elements that are inside and outside of the coupling surface are processed. The default setting of INSIDE is appropriate for most of the problems. In the majority of analyses, the Eulerian material flows around the outside of the coupling surface. Therefore, the Eulerian elements that fall within the coupling surface do not contain material. For some specific applications, such as airbag inflation, the Eulerian material (gas) is completely contained within the coupling surface. In these cases, the “COVER” definition should be set to OUTSIDE.
5. By default the fast coupling algorithm is used. The algorithm then used is substantially faster than the general coupling. The restriction is that you cannot use an arbitrarily shaped Euler mesh with the fast coupling algorithm. All element faces of the Euler mesh must have their normal pointing in any of the three basic coordinate directions. If you want to use the general coupling algorithm, you can define the parameter “DYPARAM,FASTCOUP,NO” in the input file.
6. The friction model implemented for the coupling algorithm is a simple Coulomb friction definition. The friction coefficient  $\mu$  is defined as:

$$\mu = \mu_k + (\mu_s - \mu_k) \cdot e^{-(\beta \cdot v)}$$

where  $\mu_s$  is the static friction coefficient,  $\mu_k$  is the kinetic friction coefficient,  $\beta$  the exponential decay coefficient and  $v$  the relative sliding at the point of contact.

7. An initial gas composition is for use with the single-material hydrodynamic Euler solver and an ideal-gas equation of state (EOSGAM) only.

8. Multiple coupling surfaces are available when you associate one Eulerian domain with a single coupling surface by either using the SET1ID or the MESHID option. Note that only one of the two options may be set and will work only in combination with the fast coupling algorithm.
9. The COUP1FL option is available and valid only in combination with the fast coupling algorithm with the failure option (DYPARAM,FASTCOUP, ,FAIL). If no number is given, the default values of the surrounding variables will be used; the density (RHO) is set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
10. If an ACTIVE entry is present, its definition is ignored in case the TDEAC value is defined in combination with the fast coupling algorithm.
11. If there is only one coupling surface and no adaptive meshing is used, the HYDSTAT field can be left blank. To impose a boundary condition that matches the hydrostatic initialization, the PORHYDS entry can be used.
12. A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.
13. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here,  $\tau_w$  denotes the shear friction in an Euler element adjacent to a couple surface segment where  $\rho$  is the density and  $u$  is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment.

SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

14. If the coupling surface is a structural solid then Sol700 can automatically create a coupling surface for the solid. This coupling surface consists of the boundary faces of the solid. To activate this, a BCPROP has to be created for the solid and the BCPROP has to be used for the BSID of the COUPLE entry.
15. Option NONE requires the use of DYPARAM, AUTCOUP. For details refer to this DYPARAM.

**CPENTA****Five-Sided Solid Element Connection**

Defines the connections of a five-sided solid element with six to fifteen grid points.

**Format:**

| 1      | 2   | 3   | 4  | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|----|-----|-----|-----|-----|-----|----|
| CPENTA | EID | PID | G1 | G2  | G3  | G4  | G5  | G6  |    |
|        | G7  | G8  | G9 | G10 | G11 | G12 | G13 | G14 |    |
|        | G15 |     |    |     |     |     |     |     |    |

**Example:**

|        |     |    |   |    |    |   |     |     |  |
|--------|-----|----|---|----|----|---|-----|-----|--|
| CPENTA | 112 | 2  | 3 | 15 | 14 | 4 | 103 | 115 |  |
|        | 5   | 16 | 8 |    |    |   | 120 | 125 |  |
|        | 130 |    |   |    |    |   |     |     |  |

| Descriptor | Meaning  | Type                      | Default  |
|------------|--|---------------------------|----------|
| EID        | Element identification number.                               | 0 < Integer < 100,000,000 | Required |
| PID        | Property identification number of a PSOLID or PLSOLID entry. | Integer > 0               | Required |
| Gi         | Identification numbers of connected grid points.             | Integer $\geq 0$ or blank | Required |

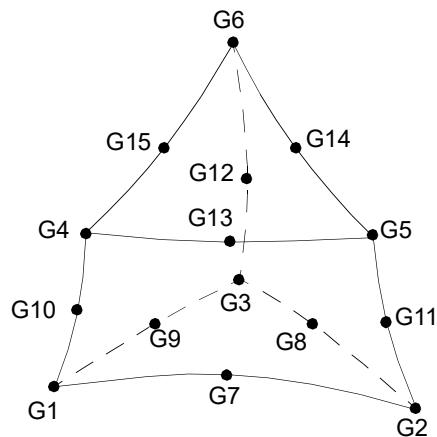


Figure 9-46 CPENTA Element Connection

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.
3. For Nastran conventional element, the edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system.
5. For nonhyperelastic elements the element coordinate system for the CPENTA element is derived accordingly. The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

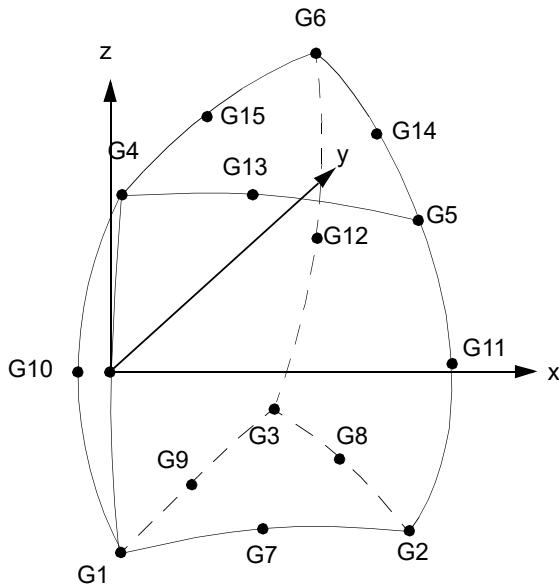


Figure 9-47 CPENTA Element Coordinate System

6. We recommend that the edge grid points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in [Item Codes, 1045](#).
8. If a CPENTA element is referenced on a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
  - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
  - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the nine edges of the element are considered straight unless:
  - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
  - For h-elements any of G7 through G15 are specified.
10. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-48](#).

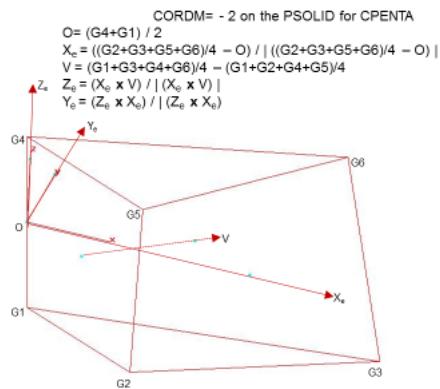


Figure 9-48      PSOLID for CPENTA

**CPYRAM****Five-Sided Solid Element Connection**

Defines the connections of the five-sided solid element with five or thirteen grid points.

**Format:**

| 1      | 2   | 3   | 4  | 5   | 6   | 7   | 8   | 9  | 10 |
|--------|-----|-----|----|-----|-----|-----|-----|----|----|
| CPYRAM | EID | PID | G1 | G2  | G3  | G4  | G5  | G6 |    |
|        | G7  | G8  | G9 | G10 | G11 | G12 | G13 |    |    |

**Example:**

|        |    |    |    |    |    |    |    |    |  |
|--------|----|----|----|----|----|----|----|----|--|
| CPYRAM | 7  | 4  | 15 | 16 | 17 | 18 | 40 | 21 |  |
|        | 22 | 23 | 24 | 31 | 32 | 33 | 34 |    |  |

| Descriptor | Meaning   | Type                      | Default  |
|------------|---|---------------------------|----------|
| EID        | Element identification number.                    | 0 < Integer < 100,000,000 | Required |
| PID        | Property identification number of a PSOLID entry. | Integer > 0               | Required |
| Gi         | Identification numbers of connected grid points.  | Integer $\geq 0$ or blank | Required |

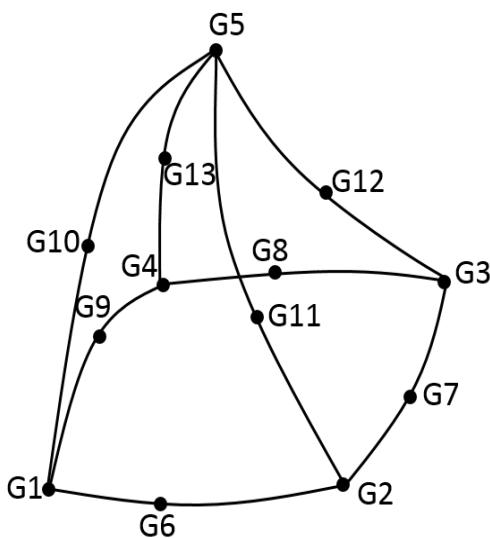


Figure 9-49 CPYRAM Element Connection

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, that is, G1, G2, G3, G4 define the quadrilateral base, G5 defines the apex, G1, G5, G10 are on the same edge, etc.
3. For a 5 noded CPYRAM element, only G1 - G5 should be provided. For a 13 noded CPYRAM element, all the corner and mid-side nodes (G1 - G13) should be provided. Note that partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. The pyramid elements use the basic coordinate system as the element coordinate system. They can use a material coordinate system defined by the CORDM field of the corresponding PSOLID entry. CORDM = 0, -1, -2 or blank is the same as the basic coordinate system.
5. For CORDM > 0, the material coordinate system is defined with respect to the initial geometry and is used to input orthotropic or anisotropic material properties and for stress output. For geometric nonlinear analysis, the material coordinate system rotates with the element. Component of stresses are output in the material coordinate system referenced by the CORDM field of PSOLID entry (CORDM > 0).

**CQUAD****Fully Nonlinear Plane Strain Element**

Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.

**Format:**

| 1     | 2   | 3   | 4  | 5             | 6  | 7  | 8  | 9  | 10 |
|-------|-----|-----|----|---------------|----|----|----|----|----|
| CQUAD | EID | PID | G1 | G2            | G3 | G4 | G5 | G6 |    |
|       | G7  | G8  | G9 | THETA or MCID |    |    |    |    |    |

**Example:**

|       |     |     |    |    |    |    |  |  |  |
|-------|-----|-----|----|----|----|----|--|--|--|
| CQUAD | 111 | 203 | 31 | 74 | 75 | 32 |  |  |  |
|-------|-----|-----|----|----|----|----|--|--|--|

| Descriptor        | Meaning  |
|-------------------|--|
| EID               | Element identification number. (0 < Integer < 100,000,000)   |
| PID               | Property identification number of a PLPLANE or PLCOMP entry. (Integer > 0)   |
| G1, G2,<br>G3, G4 | Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)  |
| G5, G6,<br>G7, G8 | Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer $\geq 0$ or blank)   |
| G9                | Identification number of center grid point. Optional. (Integer $\geq 0$ or blank)  |
| THETA             | Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=PSTRS or PLSTRN codes, THETA is measured relative to the line defined from G1-G2. For PSHLN2 or PLCOMP BEHi=COMPS code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0)   |
| MCID              | Material coordinate system identification number MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis |

| Descriptor | Meaning  |
|------------|--|
| CORD1C,    | r-axis of MCID the coordinate is projected onto shell surface through the  |
| CORD2C     | element center and the material angle is measured from the G1-G2 line to   |
| CORD1S,    | the to the projected r-axis  |
| CORD2S     |  |
|            | For PSHLN2 (or PLCOMP) BEHi=COMPS code the MCID value on the element connection entry will be ignored. (Integer $\geq 0$ ; if blank, then THETA = 0.0 is assumed.) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G9 must be numbered as shown and must lie on a plane. G5 - G9 are Optional data for any or all four grid points for Nastran conventional element only.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CQUADFD element name in [Item Codes, 1045](#).
5. Stresses and strains are output in the coordinate system identified by the CID field of the PLPLANE entry.

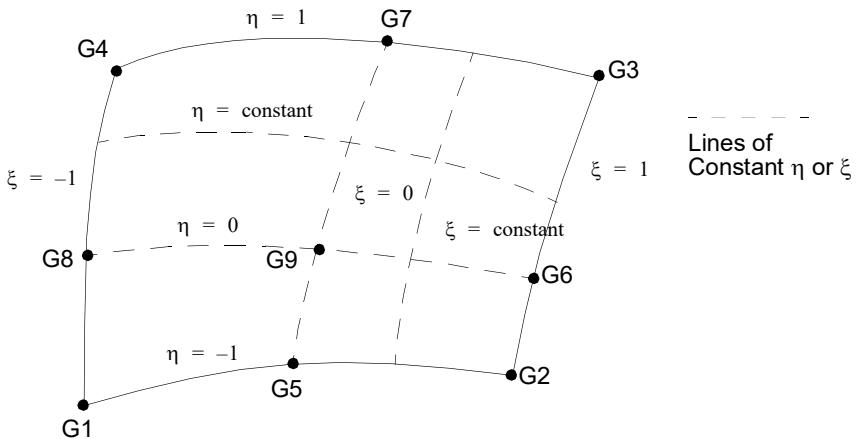


Figure 9-50 CQUAD Element Coordinate System

## CQUAD4

## Quadrilateral Plate Element Connection

Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

**Format:**

| 1      | 2   | 3     | 4  | 5  | 6  | 7  | 8             | 9     | 10 |
|--------|-----|-------|----|----|----|----|---------------|-------|----|
| CQUAD4 | EID | PID   | G1 | G2 | G3 | G4 | THETA or MCID | ZOFFS |    |
|        |     | TFLAG | T1 | T2 | T3 | T4 |               |       |    |

**Example:**

|        |     |     |      |      |      |      |     |     |  |
|--------|-----|-----|------|------|------|------|-----|-----|--|
| CQUAD4 | 111 | 203 | 31   | 74   | 75   | 32   | 2.6 | 0.3 |  |
|        |     |     | 1.77 | 2.04 | 2.09 | 1.80 |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0; Default = EID)  |
| Gi         | Grid point identification numbers of connection points.<br>(Integers > 0, all unique.)  |
| THETA      | Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. See <a href="#">Figure 9-52</a> . See Remark 10. (Real; Default = 0.0)  |
| MCID       | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R,    x -axis of MCID the coordinate is projected onto shell surface and the<br>CORD2R    material angle is measured from the G1-G2 line to the to the projected x-axis<br><br>CORD1C,    r-axis of MCID the coordinate is projected onto shell surface through the<br>CORD2C    element center and the material angle is measured from the G1-G2 line<br>CORD1S,    to the to the projected r-axis<br>CORD2S<br><br>Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 10. (Integer $\geq$ 0; If blank, then THETA = 0.0 is assumed.) |

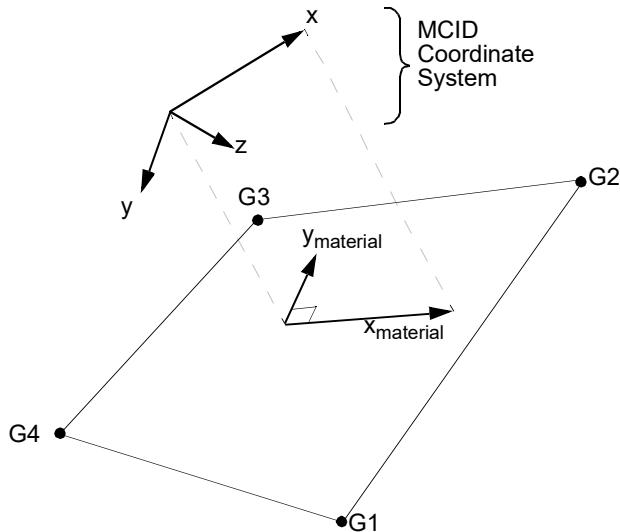


Figure 9-51 MCID Coordinate System Definition

|       |   |
|-------|---|
| ZOFFS | Offset from the surface of grid points to the element reference plane. ZOFFS is ignored for hyperelastic elements. See Remark 6. (Real)   |
| TFLAG | An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)  |
| Ti    | Membrane thickness of element at grid points G1 through G4. If "TFLAG" is zero or blank, then Ti are actual user specified thicknesses. See Remark 4. for default. (Real $\geq 0.0$ or blank, not all zero.) If "TFLAG" is one, then the Ti are fractions relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements. |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than  $180^\circ$ .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

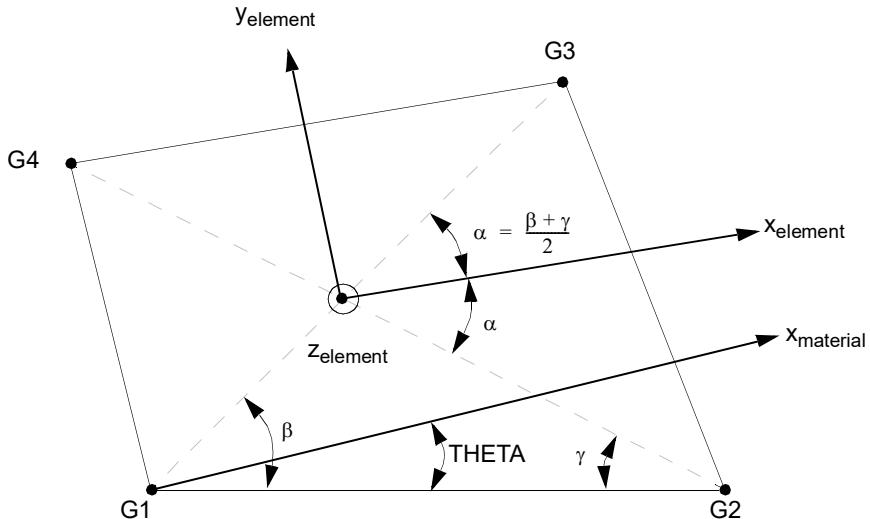


Figure 9-52 CQUAD4 Element Geometry and Coordinate Systems

5. The reference coordinate system for the output of stress, strain and element force depends on the element type.
    - For CQUAD4 elements which are not p-elements and not hyperelastic, the reference coordinate system is the default for output is the element coordinate system. See PARAM,OMID for output in the material system.
    - For CQUAD4 elements referenced by a PSET or PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system. See the Bulk Data entry, [OUTRCV, 2662](#) for user defined output coordinate systems.
    - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
  6. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.
- Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry [MDLPRM,OFFDEF,LROFF](#). For options of offsets, please refer to the Bulk Data entry [MDLPRM,OFFDEF,option](#).

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

7. For finite deformation hyperelastic analysis, the plot codes are given by the CQUADFD element name in [Item Codes, 1045](#).
8. If a CQUAD4 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
  - If a curved edge of a p-element is shared by an h-element CQUAD4, the geometry of the edge is ignored and set straight.
9. By default, all of the four edges of the element are considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE or FEFACE entry.
10. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
11. For RC network solver in thermal analysis, the ZOFFS is ignored.
12. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

**CQUAD8****Curved Quadrilateral Shell Element Connection**

Defines a curved quadrilateral shell or plane strain element with eight grid points.

**Format:**

| 1      | 2     | 3   | 4  | 5  | 6  | 7  | 8             | 9     | 10 |
|--------|-------|-----|----|----|----|----|---------------|-------|----|
| CQUAD8 | EID   | PID | G1 | G2 | G3 | G4 | G5            | G6    |    |
|        | G7    | G8  | T1 | T2 | T3 | T4 | THETA or MCID | ZOFFS |    |
|        | TFLAG |     |    |    |    |    |               |       |    |

**Example:**

|        |     |    |       |       |       |      |     |     |  |
|--------|-----|----|-------|-------|-------|------|-----|-----|--|
| CQUAD8 | 207 | 3  | 31    | 33    | 73    | 71   | 32  | 51  |  |
|        | 53  | 72 | 0.125 | 0.025 | 0.030 | .025 | 30. | .03 |  |

| Descriptor     | Meaning  |
|----------------|--|
| EID            | Element identification number. (0 < Integer < 100,000,000)   |
| PID            | Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0)  |
| G1, G2, G3, G4 | Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)  |
| G5, G6, G7, G8 | Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank).<br>Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.                                    |
| Ti             | Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. See Remark 4. for default. (Real $\geq 0.0$ or blank, not all zero.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.                       |
| THETA          | Material property orientation angle in degrees. See Figure 9-53. THETA is ignored for hyperelastic elements. See Remark 10. (Real; Default = 0.0)  |
| MCID           | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R    x -axis of MCID the coordinate is projected onto shell surface and the<br>CORD2R    material angle is measured from the G1-G2 line to the to the projected<br>x-axis |

| Descriptor                             | Meaning  |
|--|--|
| CORD1C,<br>CORD2C<br>CORD1S,<br>CORD2S | r-axis of MCID the coordinate is projected onto shell surface through the element center and the material angle is measured from the G1-G2 line to the to the projected r-axis<br><br>(see Remark 3.) MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 10. (Integer $\geq 0$ ; if blank, then THETA = 0.0 is assumed.) |
| ZOFFS                                  | Offset from the surface of grid points to the element reference plane. See Remark 6.<br>ZOFFS is ignored for hyperelastic elements. (Real)   |
| TFLAG                                  | An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)   |

### Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G8 must be numbered as shown in [Figure 9-53](#).
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between  $x_{\text{material}}$  and the line of constant  $\eta$ .

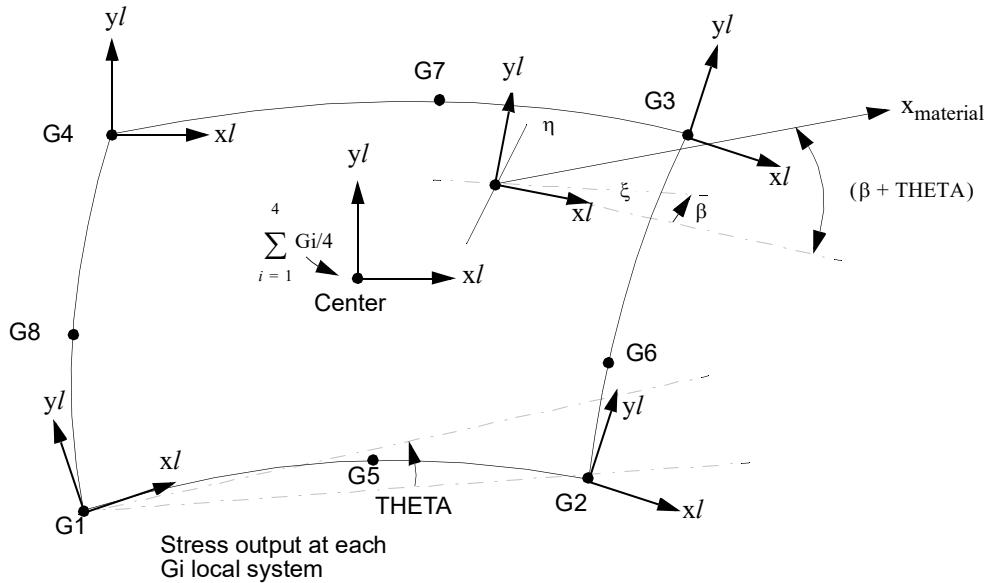
The definition of the material coordinate system by projection is used to calculate an angle THETA. Please note that since  $x_i$  changes directions throughout the element based on element shape, the material coordinate system varies similarly. Because of this an orthotropic or anisotropic material will cause the CQUAD8's stiffness to be biased by both it's shape and grid ordering. Use the QUAD4 element if a constant material coordinate system direction is desired with orthotropic and anisotropic materials.

4. T1, T2, T3 and T4 are optional. If they are not supplied and no TFLAG, then T1 through T4 will be set to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge. If the edge point is located at the quarter point, the program may fail with a divide-by-zero error or the calculated stresses will be meaningless.
6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

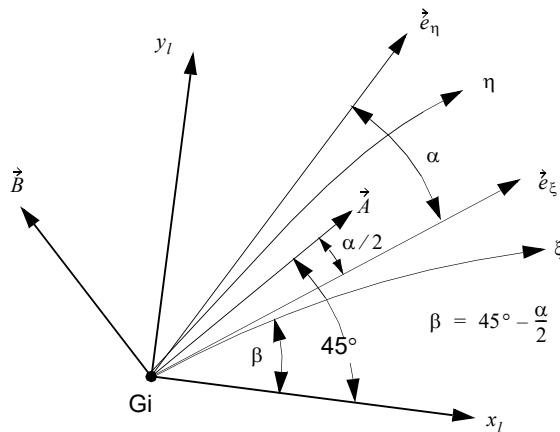
For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces incorrect. A User Warning Message is printed, and a CQUAD4 element is recommended instead. If the element is hyperelastic, then it is processed identically to the hyperelastic CQUAD4 element.
8. For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by  $x_l$  and  $y_l$  in [Figure 9-53](#). However, for hyperelastic elements the stress and strain are output in the coordinate system identified by the CID field on the PLPLANE entry.
9. For hyperelastic elements the plot codes are specified under the CQUADFD element name in [Item Codes, 1045](#).



where

- $\dot{\epsilon}_\eta$  is tangent to  $\eta$  at  $G_i$
- $\dot{\epsilon}_\xi$  is tangent to  $\xi$  at  $G_i$
- $\vec{A}$  is formed by bisection of  $\dot{\epsilon}_\eta$  and  $\dot{\epsilon}_\xi$
- $\vec{B}$  and  $\vec{A}$  are perpendicular



$y_l$  is formed by bisection of  $\vec{A}$  and  $\vec{B}$

$x_l$  is perpendicular to  $y_l$

Figure 9-53 CQUAD8 Element Geometry and Coordinate Systems

10. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
11. For RC network solver in thermal analysis, the ZOFFS is ignored.
12. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

## CQUADR

## Quadrilateral Plate Element Connection

Defines an isoparametric membrane and bending quadrilateral plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CTRIAR element.

Format:

| 1      | 2   | 3     | 4  | 5  | 6  | 7  | 8             | 9     | 10 |
|--------|-----|-------|----|----|----|----|---------------|-------|----|
| CQUADR | EID | PID   | G1 | G2 | G3 | G4 | THETA or MCID | ZOFFS |    |
|        |     | TFLAG | T1 | T2 | T3 | T4 |               |       |    |

Example:

|        |    |     |      |      |      |      |     |  |  |
|--------|----|-----|------|------|------|------|-----|--|--|
| CQUADR | 82 | 203 | 31   | 74   | 75   | 32   | 2.6 |  |  |
|        |    |     | 1.77 | 2.04 | 2.09 | 1.80 |     |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of a PSHELL, PCOMP or PCOMPEntry. (Integer > 0; Default = EID)   |
| Gi         | Grid point identification numbers of connection points. (Integers > 0, all unique)  |
| THETA      | Material property orientation angle in degrees. See <a href="#">Figure 9-55</a> . (Real; Default = 0)   |
| MCID       | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R,    x -axis of MCID the coordinate is projected onto shell surface and the<br>CORD2R    material angle is measured from the G1-G2 line to the to the projected<br>x-axis<br><br>CORD1C,    r-axis of MCID the coordinate is projected onto shell surface through the<br>CORD2C    element center and the material angle is measured from the G1-G2 line<br>CORD1S,    to the to the projected r-axis<br>CORD2S |
|            | Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; If blank, then THETA = 0.0 is assumed.)   |
| ZOFFS      | Offset from the surface of grid point to the element plane. See Remark 8.   |

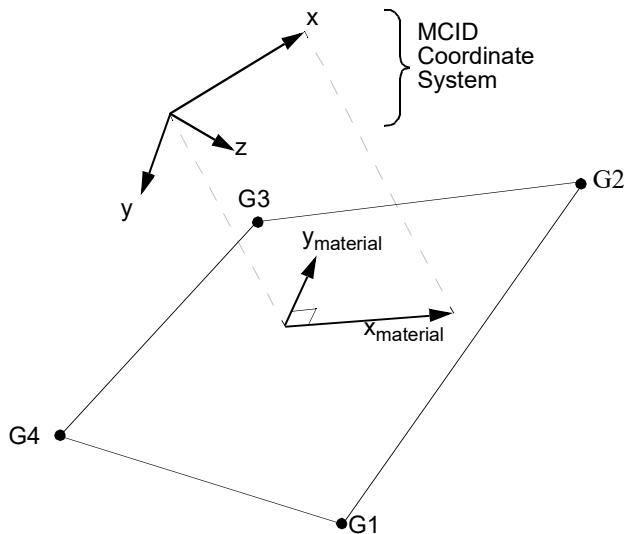


Figure 9-54 MCID Coordinate System Definition

|       |  |
|-------|--|
| TFLAG | An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)   |
| Ti    | Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements. |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All the interior angles must be less than  $180^\circ$ .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.
5. Stresses and strains are output in the element coordinate system at the centroid and grid points G1 through G4.
6. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4entry \(SORL option\)](#) and the [Consistent Surface and Edge Loads](#) (Ch. 3) in the *MSC Nastran Reference Guide* for additional information.

7. The CTRIAR element is the triangular companion to the CQUADR element and should be used instead of CTRIA3 or CTRIA6.

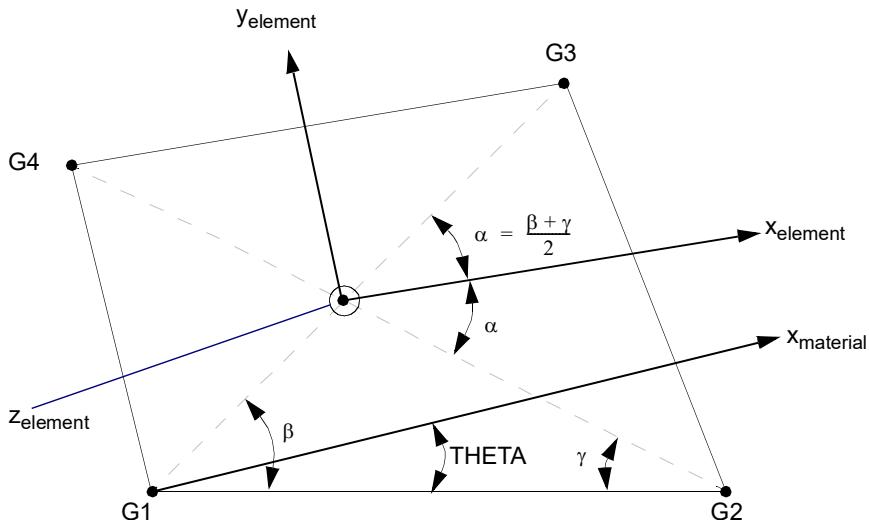


Figure 9-55 CQUADR Element Geometry and Coordinate Systems

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

9. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

**CQUADX****Axisymmetric Quadrilateral Element (Fully Nonlinear or Linear Harmonic)**

Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) analysis or a linear harmonic or rotordynamic analysis. The element has between four and nine grid points.

**Format:**

| 1      | 2   | 3   | 4  | 5             | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|---------------|----|----|----|----|----|
| CQUADX | EID | PID | G1 | G2            | G3 | G4 | G5 | G6 |    |
|        | G7  | G8  | G9 | THETA or MCID |    |    |    |    |    |

**Example:**

|        |     |     |    |    |    |    |  |  |  |
|--------|-----|-----|----|----|----|----|--|--|--|
| CQUADX | 111 | 203 | 31 | 74 | 75 | 32 |  |  |  |
|--------|-----|-----|----|----|----|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ ). See Remark 1.  |
| PID        | Property identification number of a PLPLANE or PAXSYMH or PLCOMP entry. ( $\text{Integer} > 0$ ). See Remark 2.   |
| G1, G2     | Identification numbers of connected corner grid points. Required data for all four grid points. ( $\text{Unique Integers} > 0$ ). See Remark 3., 6., 7.   |
| G3, G4     |   |
| G5, G6     | Identification numbers of connected edge grid points. Optional data for any or all four grid points. ( $\text{Integer} \geq 0$ or blank). See Remark 3., 4., 6., 7.   |
| G7, G8     |   |
| G9         | Identification number of center grid point. Optional. ( $\text{Integer} \geq 0$ or blank). Not used for linear harmonic elements. See Remark 3., 6., 7.   |
| THETA      | Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. For PSHLN2 or PLCOMP BEHi=AXCOMP code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0). THETA is ignored if PID refers to a PAXSYMH entry.  |
| MCID       | Material coordinate system identification number, MCID is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. For PSHLN2 BEHi=AXSOLID the resulting angle is measured relative to the R axis of the element. For PSHLN2 (or PLCOMP) BEHi=AXCOMP code the MCID value on the element connection entry will be ignored. (Integer $\geq 0$ ; If blank, then THETA = 0.0 is assumed.) |

**Remarks:**

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. If PID refers to a PLPLANE or PLCOMP entry, CQUADX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CQUADX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-56](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CQUADXF D element name in [Item Codes, 1045](#).
6. All Gi must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is multiplied by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50 \text{ N}$$

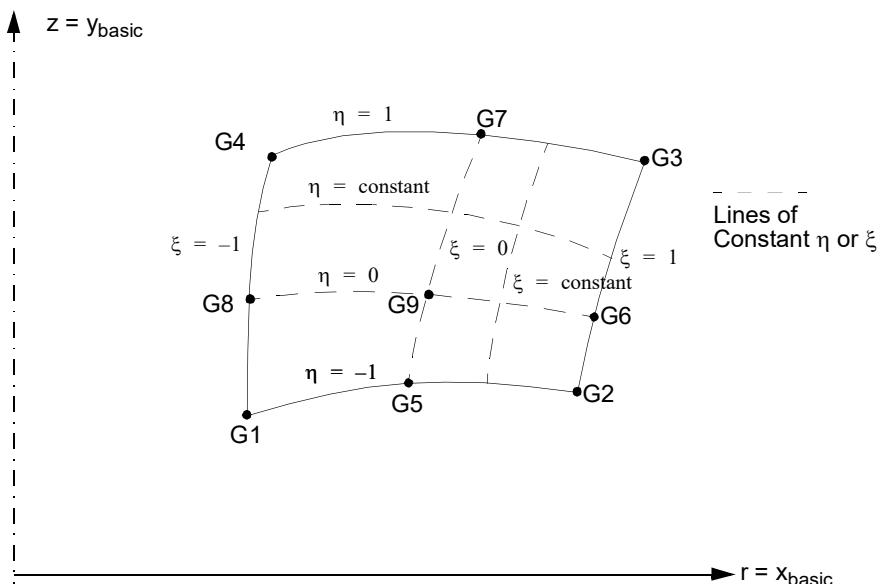


Figure 9-56 CQUADX Element Coordinate System

**CRAC2D****Two-Dimensional Crack Tip Element**

Defines a two-dimensional crack tip element.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CRAC2D | EID | PID | G1  | G2  | G3  | G4  | G5  | G6  |    |
|        | G7  | G8  | G9  | G10 | G11 | G12 | G13 | G14 |    |
|        | G15 | G16 | G17 | G18 |     |     |     |     |    |

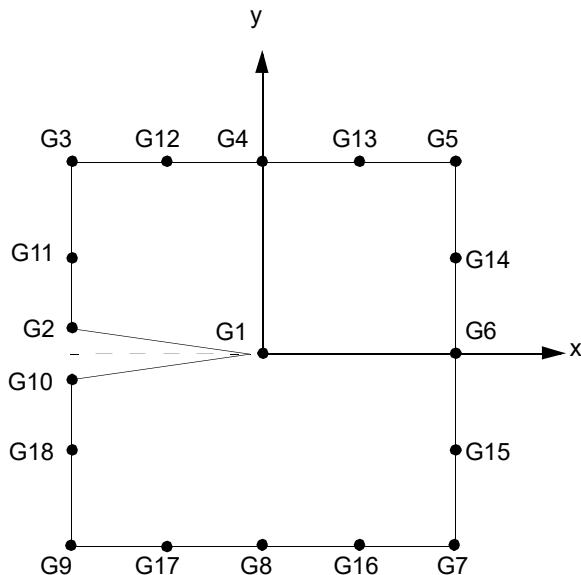
**Example:**

|        |     |     |    |    |   |    |    |    |  |
|--------|-----|-----|----|----|---|----|----|----|--|
| CRAC2D | 114 | 108 | 2  | 5  | 6 | 8  | 7  | 11 |  |
|        | 12  | 14  | 16 | 17 |   | 20 | 22 |    |  |

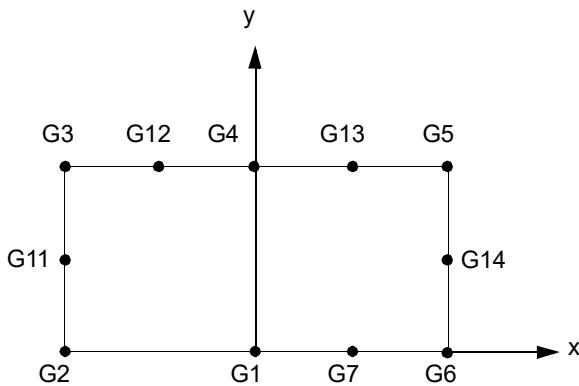
| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PRAC2D entry. (Integer > 0)  |
| Gi         | Grid point identification numbers of connection points. (Integer $\geq 0$ ; G11 through G18 may be blank.) |

**Remarks:**

1. CRAC2D is a dummy element and requires the presence of this Bulk Data entry:
- |       |    |   |   |   |        |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|
| ADUM8 | 18 | 0 | 5 | 0 | CRAC2D |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|
2. The element should be planar. Significant deviations will produce fatal errors.
  3. Grid points G1 through G10 are required while grid points G11 through G18 are optional for the quadrilateral form of the element.
  4. The stresses and stress intensity factors are calculated assuming that G2 and G10 are coincident. Deviations from this will produce erroneous results.
  5. For the symmetric half-crack option, grid points G1 through G7 are required while grid points G11 through G14 are optional. Grid points G8 through G10 and G15 through G18 must not be present for this option.
  6. The ordering conventions for the full-crack and half-crack options are shown in [Figure 9-57](#).
  7. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-57](#) should be in the range 2.0 to 0.5.
  8. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) (p. 177) in the *MSC Nastran Reference Guide*.



(a) Full Crack Option



(b) Symmetric Half-Crack Option

Figure 9-57 CRAC2D Element Connection for Full and Symmetric Options

**CRAC3D****Three-Dimensional Crack Tip Element**

Defines a three-dimensional crack tip element.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| CRAC3D | EID | PID | G1  | G2  | G3  | G4  | G5  | G6  |    |
|        | G7  | G8  | G9  | G10 | G11 | G12 | G13 | G14 |    |
|        | G15 | G16 | G17 | G18 | G19 | G20 | G21 | G22 |    |
|        | G23 | G24 | G25 | G26 | G27 | G28 | G29 | G30 |    |
|        | G31 | G32 | G33 | G34 | G35 | G36 | G37 | G38 |    |
|        | G39 | G40 | G41 | G42 | G43 | G44 | G45 | G46 |    |
|        | G47 | G48 | G49 | G50 | G51 | G52 | G53 | G54 |    |
|        | G55 | G56 | G57 | G58 | G59 | G60 | G61 | G62 |    |
|        | G63 | G64 |     |     |     |     |     |     |    |

**Example:**

|        |     |     |     |     |     |     |     |     |  |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| CRAC3D | 113 | 101 | 2   | 5   | 7   | 8   | 4   | 10  |  |
|        | 11  | 14  | 15  | 17  |     | 3   | 6   | 9   |  |
|        | 12  |     | 16  |     | 102 | 105 | 107 | 108 |  |
|        | 104 | 110 | 111 | 114 | 115 | 117 |     | 103 |  |
|        | 106 | 109 | 112 |     | 116 |     | 202 | 205 |  |
|        | 207 | 208 | 204 | 210 | 211 | 214 | 215 | 217 |  |
|        | 225 | 226 |     |     |     |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )               |
| PID        | Property identification number of a PRAC3D entry. ( $\text{Integer} < 0$ )          |
| Gi         | Grid point identification numbers of connection points. ( $\text{Integer} \geq 0$ ) |

**Remarks:**

1. CRAC3D is a dummy element and requires the presence of this Bulk Data entry:

|       |    |   |   |   |        |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|
| ADUM9 | 64 | 0 | 6 | 0 | CRAC3D |  |  |  |  |
|-------|----|---|---|---|--------|--|--|--|--|

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. This element, including grid point numbering conventions, is shown in [Figure 9-58](#) and [Figure 9-59](#). Grid points G1 through G10, and G19 through G28 are required; midside and surface grid points G11 through G18, G29 through G36, and G37 through G64 are optional. Either all or none of grid points G37 through G46 should be present. A fatal error message will be issued for partial connectivity.
4. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-58](#) should be in the range 2.0 to 0.5.
5. For the symmetric half-crack option Grid Points G1 through G7, and G19 through G25 are required, whereas grid points G11 through G14, G29 through G32, and G37 through G42 are optional. Grid points G8 through G10, G15 through G18, G26 through G28, G33 through G36, G43 through G46, G51 through G55, and G60 through G64 should not be specified to invoke this option.
6. It is recommended that both the faces (formed by grid points G2 through G18 and grid points G20 through G36) and the midplane (formed by grid points G37 through G46 and grid points G37 through G46) be planar. It is also recommended that midside grid points G37 through G46 be located within the middle third of the edges.
7. The midside nodes on both the faces should be defined in pairs. For example, if grid point G11 is not defined, then grid point G29 should not be defined and vice versa.
8. The stresses and stress intensity factors are calculated with the assumptions that grid points G2 and G10, G20 and G28, and G38 and G46 are coincident. Deviation from this condition will produce erroneous results.
9. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) (p. 177) in the *MSC Nastran Reference Guide*.
10. As depicted in [Figure 9-58](#) and [Figure 9-59](#), the element is a right-handed element. Thus define the vectors  $\overline{G1G9}$  and  $\overline{G1G3}$ , then the cross-product  $\overline{G1G9} \times \overline{G1G3}$  points to the face defined by G19, G20, ...

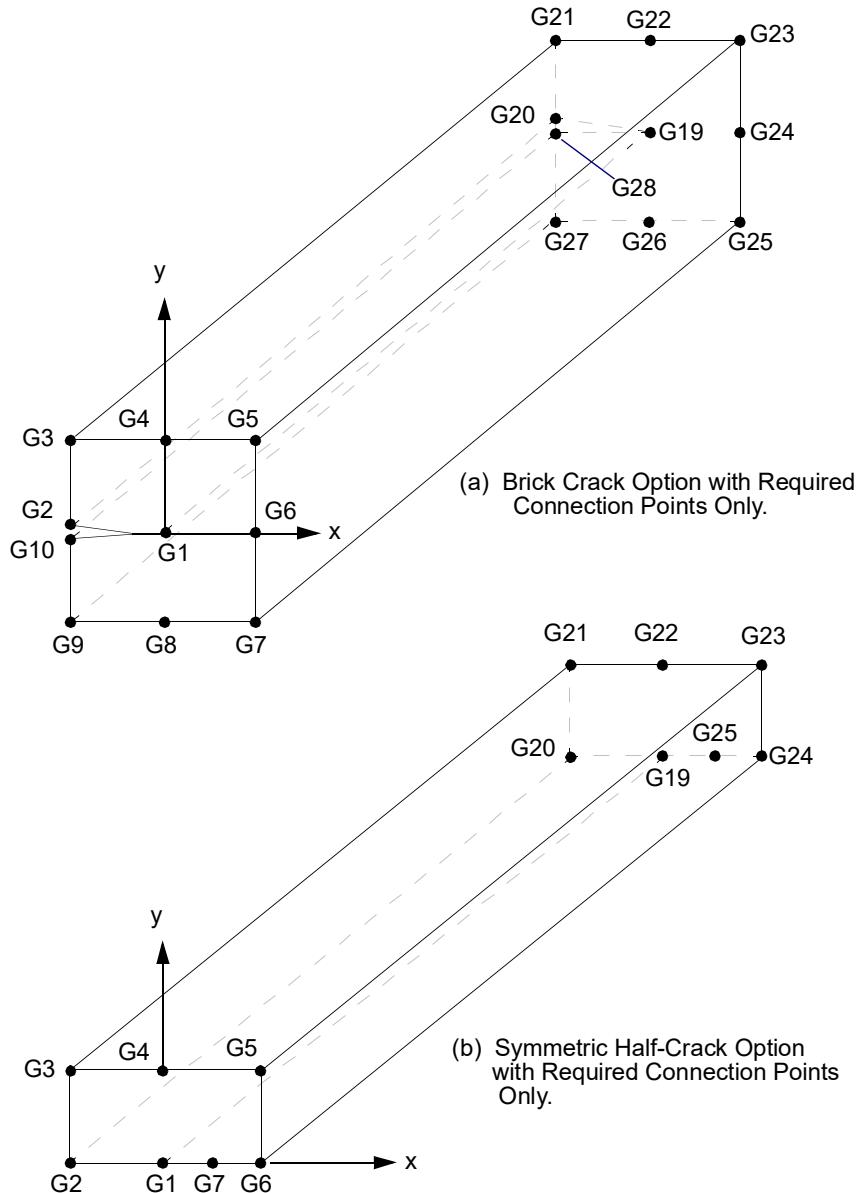


Figure 9-58 CRAC3D Solid Crack Tip Element with Required Connection Points Only

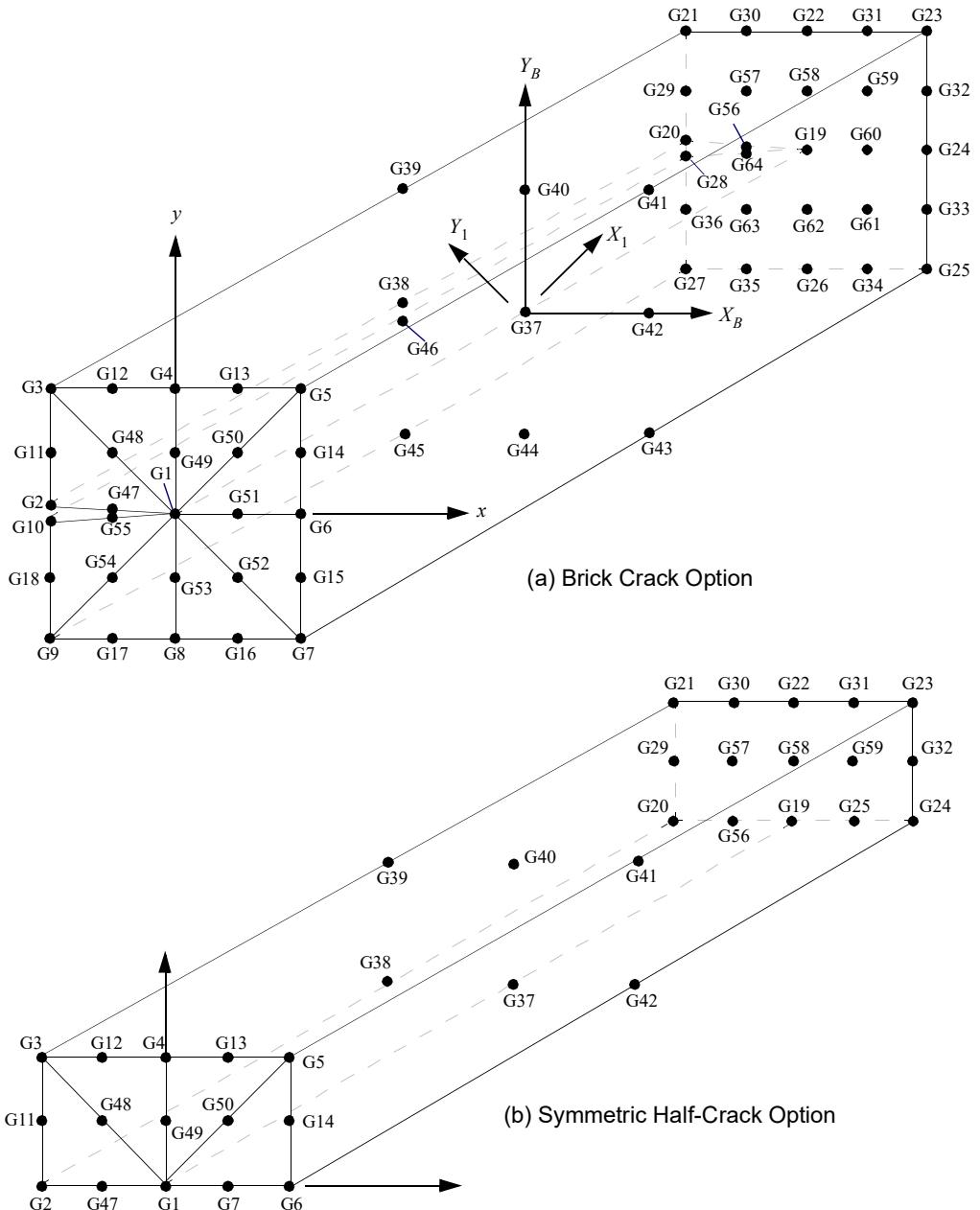


Figure 9-59 CRAC3D Solid Crack Tip Element with All Connection Points

**CREEP****Creep Characteristics**

Defines creep characteristics based on experimental data or known empirical creep law. This entry will be activated if a MAT1, MAT2, or MAT9 entry with the same MID is used and the NLPARM entry is prepared for creep analysis. The creep formulation is principally suited for isotropic materials and, in general, when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.

**Format:**

| 1     | 2    | 3  | 4   | 5    | 6     | 7     | 8     | 9      | 10 |
|-------|------|----|-----|------|-------|-------|-------|--------|----|
| CREEP | MID  | T0 | EXP | FORM | TIDKP | TIDCP | TIDCS | THRESH |    |
|       | TYPE | a  | b   | c    | d     | e     | f     | g      |    |

**Example:**

|       |     |         |       |         |        |        |        |     |  |
|-------|-----|---------|-------|---------|--------|--------|--------|-----|--|
| CREEP | 8   | 1100.   |       | CRLAW   |        |        |        |     |  |
|       | 121 | 6.985-6 | 2.444 | 7.032-4 | 0.1072 | 6.73-9 | 0.1479 | 3.0 |  |

| Descriptor                | Meaning  |
|---------------------------|--|
| MID                       | Material identification number of a MAT1, MAT2, or MAT9 entry. (Integer > 0)   |
| T0                        | Reference temperature at which creep characteristics are defined. See Remark 2. (Real; Default = 0.0)  |
| EXP                       | Temperature-dependent term, $e^{(-\Delta H/(R \cdot T0))}$ , in the creep rate expression. See Remark 2. (0.0 < Real $\leq$ 1.0; Default = 1.0E-9)   |
| FORM                      | Form of the input data defining creep characteristics. (Character: “CRLAW” for empirical creep law, or “TABLE” for tabular input data of creep model parameters.)                            |
| TIDKP,<br>TIDCP,<br>TIDCS | Identification number of a TABLES1 entry, which defines the creep model parameters $K_p(\sigma)$ , $C_p(\sigma)$ , and $C_s(\sigma)$ , respectively. See Remarks 3. through 4. (Integer > 0) |
| THRESH                    | Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young’s modulus. (0.0 < Real < 1.0E-3; Default = 1.0E-5)            |
| TYPE                      | Identification number of the empirical creep law type. See Remark 1. (Integer: 111, 112, 121, 122, 211, 212, 221, 222, or 300)   |
| a through g               | Coefficients of the empirical creep law specified in TYPE. Continuation should not be specified if FORM = “TABLE”. See Remark 1. (Real)  |

**Remarks:**

- Two classes of empirical creep law are available.

Creep Law Class 1

The first creep law class is expressed as:

$$\dot{\epsilon}^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t \quad (9-3)$$

Parameters  $A(\sigma)$ ,  $R(\sigma)$ , and  $K(\sigma)$  are specified in the following form, as recommended by Oak Ridge National Laboratory:

| Parameter   | Function 1                   | Digit | Function 2     | Digit |
|-------------|------------------------------|-------|----------------|-------|
| $A(\sigma)$ | $a\sigma^b$                  | i=1   | $ae^{b\sigma}$ | i=2   |
| $R(\sigma)$ | $ce^{d\sigma}$               | j=1   | $c\sigma^d$    | j=2   |
| $K(\sigma)$ | $e \cdot [\sinh(f\sigma)]^g$ | k=1   | $ee^{f\sigma}$ | k=2   |

TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines  $A(\sigma) = a\sigma^b$ ,  $R(\sigma) = c\sigma^d$ , and  $K(\sigma) = ee^{f\sigma}$ .

## Creep Law Class 2

The second creep law class (TYPE=300) is expressed as:

$$\dot{\epsilon}^c(\sigma, t) = a\sigma^b t^d \quad (9-4)$$

where the values of b and d must be defined as follows:

The above expression is determined by curve fitting using a Newton-Raphson procedure based on the expressions:

$$\lambda = C_p / K_{pi} = \left( \frac{\bar{\epsilon}^c}{a\bar{\sigma}^b} \right)^{1/d} \quad (9-5)$$

where  $\bar{\sigma}$  is the effective stress and the nonlinear equation:

$$\lambda^2 (e^{t_0/\lambda} - 1) - \lambda t_0 - t_0^2 / d = 0$$

Then the creep model parameters are determined by:

$$\frac{1}{C_s} = \frac{\bar{\epsilon}^c d}{t_0 \bar{\sigma}} \left[ 1 - \frac{\lambda(1-d)}{t_0} \right]$$

$$\frac{1}{K_p} = \left( \frac{\lambda}{t_0} \right)^2 \frac{\bar{\epsilon}^c}{\bar{\sigma}} (1-d)d - \frac{t_0}{C_s} + \frac{\bar{\epsilon}^c}{\bar{\sigma}}$$

$$C_p = \lambda K_p$$

To keep the solution of these expressions stable, the values of b and d are recommended to be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

- Creep law coefficients a through g are usually determined by least squares fit of experimental data, obtained under a constant temperature. This reference temperature at which creep behavior is characterized must be specified in the T0 field if the temperature of the structure is different from this reference temperature. The conversion of the temperature input ( $^{\circ}\text{F}$  or  $^{\circ}\text{C}$ ) to  $^{\circ}\text{K}$  (degrees Kelvin) must be specified in the PARAM,TABS entry as follows:

PARAM,TABS,273.16 (If Celsius is used.)

PARAM,TABS,459.69 (If Fahrenheit is used.)

When the correction for the temperature effect is required, the temperature distribution must be defined in the Bulk Data entries (TEMP, TEMPP1 and/or TEMPRB), which are selected by the Case Control command TEMP(LOAD) = SID within the subcase.

From the thermodynamic consideration, the creep rate is expressed as:

$$\dot{\varepsilon}^c = \dot{\varepsilon}_A (e^{-\Delta H / RT}) \quad (9-6)$$

where:

$\Delta H$  = energy of activation

$R$  = gas constant ( $= 1.98 \text{ cal/mole } ^{\circ}\text{K}$ )

$T$  = absolute temperature ( $^{\circ}\text{K}$ )

$\dot{\varepsilon}_A$  = strain/sec per activation

If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by a factor

$$\frac{\dot{\varepsilon}^c}{\dot{\varepsilon}_o} = \text{EXP}^{\left(\frac{T_0}{T} - 1\right)} \quad (9-7)$$

where:

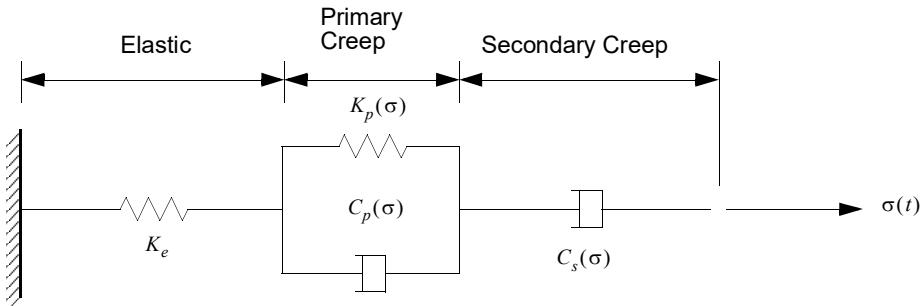
$\dot{\varepsilon}^c$  = corrected creep rate

$\dot{\varepsilon}_o$  = creep rate at T0

$\text{EXP}^{\left(\frac{T_0}{T} - 1\right)}$  = correction factor

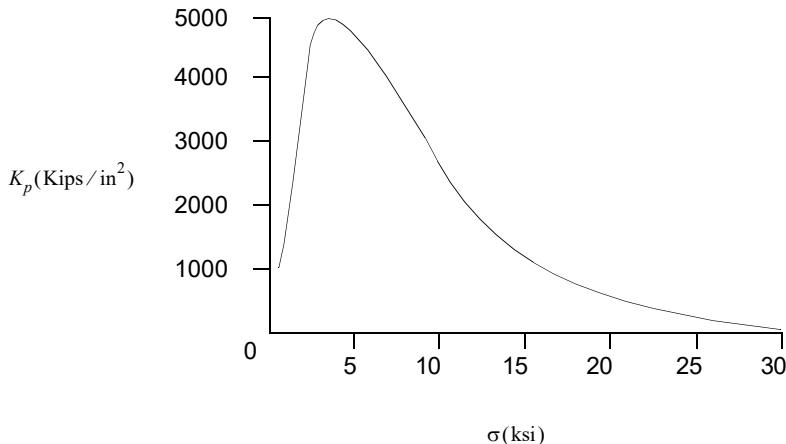
3. Creep model parameters  $K_p$ ,  $C_p$ , and  $C_s$  represent parameters of the uniaxial rheological model as shown in [Figure 9-60](#).

Tabular values ( $X_i$ ,  $Y_i$ ) in the TABLES1 entry correspond to  $(\sigma_i, K_{pi})$ ,  $(\sigma_i, C_{pi})$ , and  $(\sigma_i, C_{si})$  for the input of  $K_p$ ,  $C_p$ , and  $C_s$ , respectively. For linear viscoelastic materials, parameters  $K_p$ ,  $C_p$  and  $C_s$  are constant and two values of  $\sigma_i$  must be specified for the same value of  $K_{pi}$ ,  $C_{pi}$  and  $C_{si}$ .

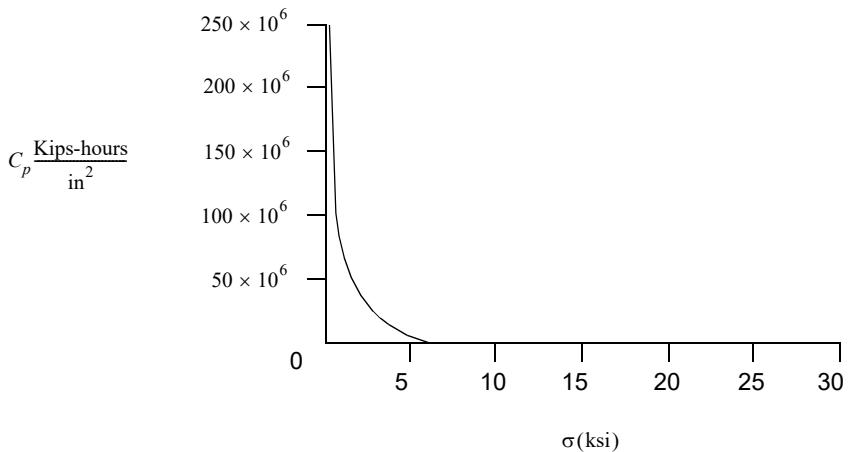
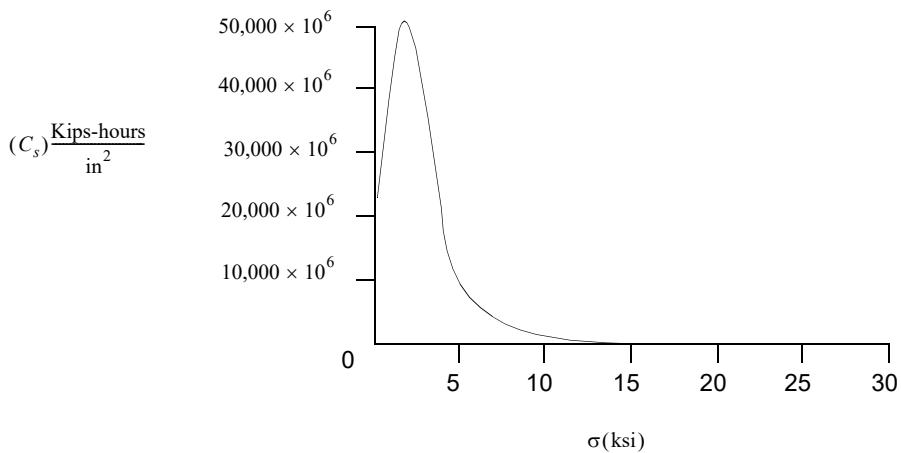


**Figure 9-60** CREEP Parameter Idealization

Creep model parameters, as shown in [Figure 9-61](#) through [Figure 9-63](#), must have positive values. If the table look-up results in a negative value, the value will be reset to zero and a warning message (TABLE LOOK-UP RESULTS IN NEGATIVE VALUE OF CREEP MODEL PARAMETER IN ELEMENT ID=\*\*\*\*\*) will be issued.



**Figure 9-61**  $K_p$  Versus  $\sigma$  Example for CREEP

Figure 9-62  $C_p$  Versus  $\sigma$  Example for CREEPFigure 9-63  $C_s$  Versus  $\sigma$  Example for CREEP

4. Creep analysis requires an initial static solution at  $t = 0$ , which can be obtained by specifying a subcase that requests an NLPARM entry with DT = 0.0.

**CROD****Rod Element Connection**

Defines a tension-compression-torsion element.

Format:

| 1    | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|------|-----|-----|----|----|---|---|---|---|----|
| CROD | EID | PID | G1 | G2 |   |   |   |   |    |

Example:

|      |    |    |    |    |  |  |  |  |  |
|------|----|----|----|----|--|--|--|--|--|
| CROD | 12 | 13 | 21 | 23 |  |  |  |  |  |
|------|----|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)                     |
| PID        | Property identification number of a PROD entry. (Integer > 0; Default = EID)   |
| G1, G2     | Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See [CONROD, 1567](#) for alternative method of rod definition.
3. Only one element may be defined on a single entry.

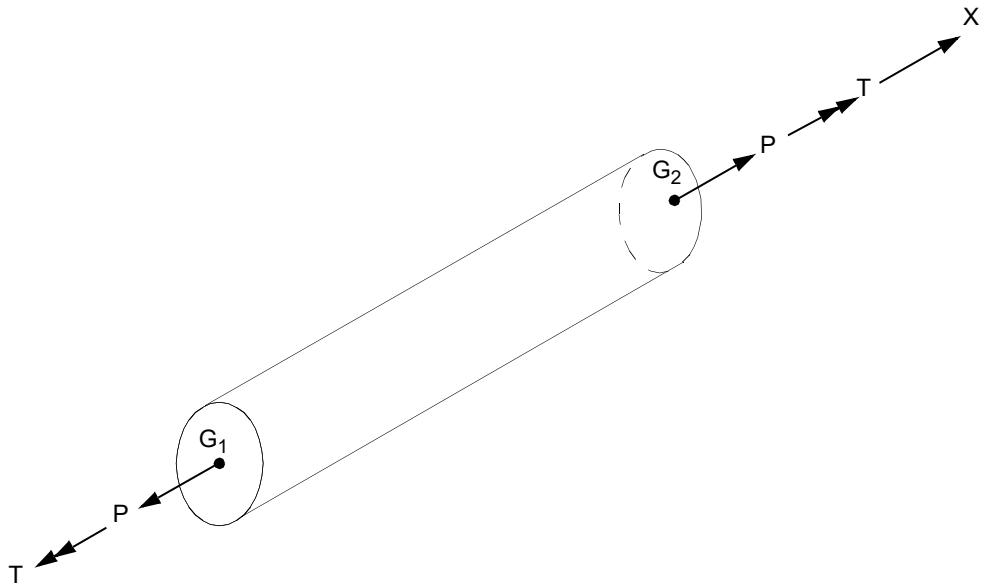


Figure 9-64 CROD Element Internal Forces and Moments

## CSEAM

## A Shell Patch SEAM Connection

Defines a SEAM connecting two surface patches.

Format:

| 1     | 2   | 3   | 4    | 5     | 6    | 7    | 8    | 9    | 10 |
|-------|-----|-----|------|-------|------|------|------|------|----|
| CSEAM | EID | PID | SMLN | CTYPE | IDAS | IDBS | IDAE | IDBE |    |
|       | GS  | GE  |      |       |      |      |      |      |    |

Alternate Format:

| 1     | 2   | 3   | 4  | 5     | 6    | 7    | 8    | 9    | 10 |
|-------|-----|-----|----|-------|------|------|------|------|----|
| CSEAM | EID | PID |    | CTYPE | IDAS | IDBS | IDAE | IDBE |    |
|       | XS  | YS  | ZS | XE    | YE   | ZE   |      |      |    |

Example:

|       |       |       |  |  |    |    |  |  |  |
|-------|-------|-------|--|--|----|----|--|--|--|
| CSEAM | 552   | 297   |  |  | 43 | 48 |  |  |  |
|       | 30422 | 77987 |  |  |    |    |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of a PSEAM entry. (Integer > 0)  |
| SMLN       | SEAM line identification. See Remark 2. (CHAR or blank)   |
| CTYPE      | Connectivity search type. (Character)<br>If CTYPE = “PSHELL”, IDAS and IDBS are property identification numbers of PSHELL’s. (Default)<br>If CTYPE = “ELEM”, IDAS and IDBS are element identification numbers.  |
| IDAS, IDBS | Used to define patch A and B or the start of patch A or B for a tailored blank. See Remark 4. (Integer > 0)<br>If CTYPE = “PSHELL”, required property id defining patches A and B. If CTYPE = “PSHELL” and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom. See Remark 6.<br>If CTYPE = “ELEM”, required element id defining patches A and B. IDAS ≠ IDBS. |
| IDAE, IDBE | Used to define the end of patch A and the end of patch B for a tailored blank. See Remark 4. (Integer ≥ 0 or blank)<br>If CTYPE = “PSHELL”, property id defining patches A and B. If CTYPE = ‘PSHELL’ and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom.<br>If CTYPE = “ELEM”, element id defining patches A and B. IDAE ≠ IDBE.                           |

| Descriptor | Meaning   |
|------------|---|
| GS, GE     | Grid ids of piercing points on patches A and B of the Start and End of the SEAM.<br>(Integer > 0) |
| XS,YS,ZS   | Location of the SEAM Start. (Real or blank)   |
| XE,YE,ZE   | Location of the SEAM End. (Real or blank)   |

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. With no embedded blanks any combination of up to eight of the acceptable characters in MSC Nastran may be employed for the SMLN entry.

The seam line will be considered continuous between each connected element and where any two elements have a common face, the faces of the resulting internal CHEXA's will be adjusted to a single common face. If a CSEAM's GS or GE is not common to the GE or GS of any other CSEAM the faces will not be adjusted.

A SMLN cannot have a branch.

3. GS and GE define the start and end points of the SEAM element. At these points and using the value W specified on the PSEAM entry, surface patches A and B are determined. Points are projected onto the surface patches A and B with the four points at end GS and the four points at end GE then used to form faces of a CHEXA element.

The auxiliary points forming the faces of the CHEXA element are then connected to the physical grids of the patches. The number of unique physical grids per patch ranges from a possibility of 6 to 64 grids.

The auxiliary points must have a projection on patches A and B, but they do not have to lie on patch A or B.

A maximum of three shell elements of patch A and three shell elements of patch B can be connected with one CSEAM element, see [Figure 9-65](#).

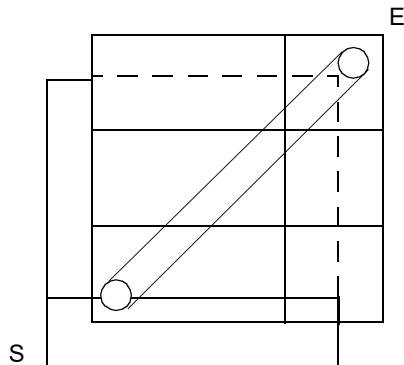


Figure 9-65 Connected Shell Elements for a CSEAM Element

#### 4. For CTYPE = ‘PSHELL’

- If patch A is uniform in thickness, then only its IDAS is needed to define it.
- If patch B is uniform in thickness, then only its IDBS is needed to define it.
- If patch A has stepped tapering, then IDAS and IDAE are used to define it.
- If patch B has stepped tapering, then IDBS and IDBE are used to define it.

#### 5. Projection Algorithms for the CSEAM Elements

Because of complex geometry, the user supplied start point GS may not have a projection SA and SB, and the end point GE may not have a projection EA and EB. Even though these four projection points are found, the program still has to find projections for the eight auxiliary points SA1, SA2, ..., EB2, and EB1 of the HEXA. The default projection strategy can be changed by overwriting the default values of the flags and parameters in the SWLDPRM Bulk Data entry.

##### a. Find Projections for SA, SB, EA, and EB

For CTYPE = “PSHELL”, the program finds the closest shell grids to GS and GE. The shell elements that are connected to these closest grids are defined as the candidate shell elements. While looping through each candidate shell element to compute the projection of GS and GE onto that element, the program always tries to get the most accurate projection. Even though a projection is found with PROJTOL > 0.0, the program still continues the projection calculations using PROJTOL=0.0. If a projection is found with PROJTOL=0.0, that shell element will be selected as the connecting element. Otherwise, the shell element that gets projection with PROJTOL>0.0 is selected as EIDSA, EIDS, EIDEA, or EIDEB. For CTYPE = “ELEM”, the above processes are skipped, because EIDSA, EIDS, EIDEA, and EIDEB have already been specified by the user.

If GSTOL > 0.0 and the distance GS-SA, GS-SB, GE-EA, or GE-EB is greater than GSTOL, a UFM 7549 is issued and the CSEAM element is rejected.

If the projection of GS or GE lies outside the shell sheet, or the connected shell elements fail the geometry check with GMCHK > 0, the program will issue a UFM and the CSEAM element will be rejected.

If GMCHK > 0, the program checks errors of CSEAM across a cutout or over a corner with elements in plane. The program also computes the angle between the shell normal vectors of EIDSA and EIDEA and the angle between the shell normal vectors of EIDS and EIDEB to check a corner with elements out of plane.

For CTYPE = “PSHELL”, if there is an error detected, the program loops back to compute the other possibility of projection until a correct connection is found or all candidate shell elements are processed. In the latter case, either UFM 7638 (the seam spans a cutout) or UFM 7667 (the seam spans a corner) will be issued. If GMCHK=2, the program also lists all candidate shell elements with their projection status for each connecting type after issuing a UFM. This will help the user to select the correct shell elements for EIDSA, EIDS, EIDEA, and EIDEB.

For CTYPE=“ELEM”, the program only checks errors and issues UFM 7638 or 7667 for the kind of error encountered. No looping back will be performed.

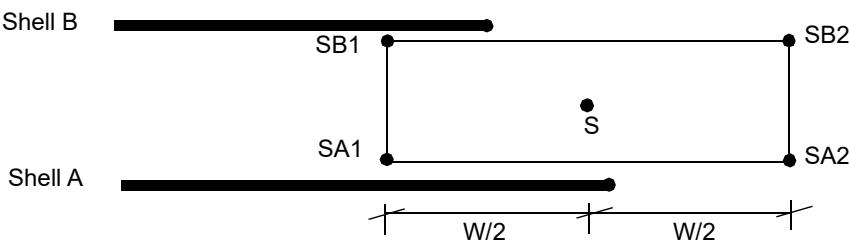
If  $GMCHK > 0$  and  $GSPROJ \geq 0.0$ , the program also computes the angle between the shell normal vectors of EIDSA and EIDS<sub>B</sub> and the angle between the shell normal vectors of EIDEA and EIDE<sub>B</sub>. A UFM 7595 is issued if the angle between the shell normal vectors is greater than GSPROJ. By default, GSPROJ = 20°, that means the shell patches A and B can be tilted relative to each other by not more than 20°.

- Find Projections of the Eight Auxiliary Points SA1, SA2, ..., EB2, and EB1

After the projections for SA, SB, EA, and EB have been found, eight auxiliary points for an internal hexagonal polygon are formed. If the GS or GE of a CSEAM element is connected to the GE or GS of another CSEAM element, then the internal HEXA elements are adjusted to a common face.

If  $GSPROJ \geq 0.0$  and the angle between the face vectors parallel to the thickness direction of the internal HEXA and the normal vector of the shell element that gets projection exceeds GSPROJ, the program will skip picking this shell element and will proceed to process next candidate shell element.

The most common error condition occurs when the seam lies on the edge of the shell patches. Under this situation, half of the seam hangs outside the shell sheets ([Figure 9-66](#)). It is required that each of the eight points has a projection. If at least one point does not have a projection and  $GSMOVE > 0$ , GS will be moved by  $W/2$ . Same algorithms apply to end E. The move will be repeated until either all projections are found or the number of moves reaches  $GSMOVE$ .



**Figure 9-66** Seam Weld at an Edge

- Error Checks by GMCHK Parameter

The GMCHK parameter specified in the SWLDPRM Bulk Data entry checks the errors of CSEAM elements across cutouts or over corners. There are three allowable values of GMCHK.

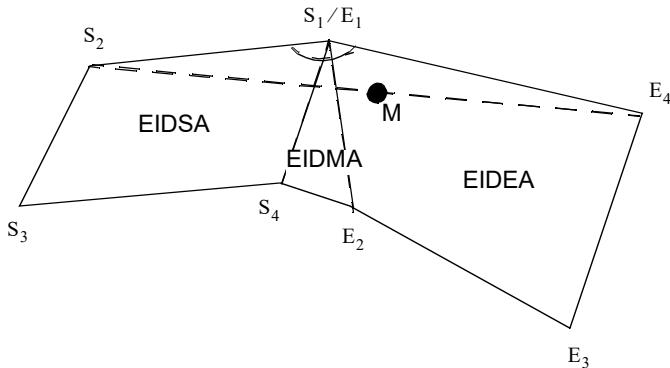
- GMCHK = 0 (Default) Do not check errors
- GMCHK = 1 Check errors
- GMCHK = 2 Check errors and output all candidate shell elements if there is an error encountered

If GMCHK is turned on, Nastran will perform the following checking while searching for the projected shell elements. Note that EIDSA is the shell element that gets projection from GS on shell A; EIDEA is the shell element that gets projection from GE on shell A. Same algorithms are applied to EIDS<sub>B</sub> and EIDE<sub>B</sub> for shell B.

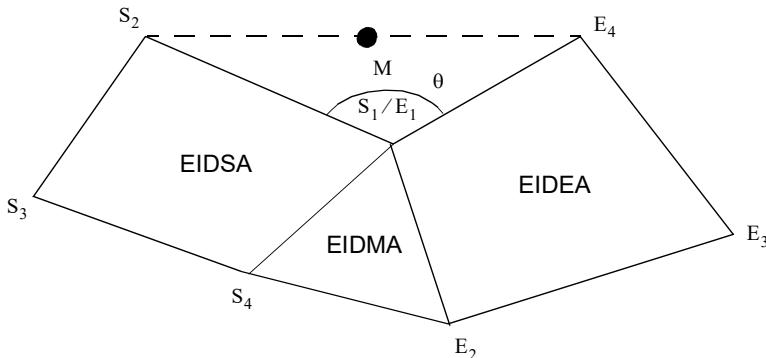
- Check the CSEAM Across a Cutout or Over a Corner with Elements in Plane
  - If EIDSA is equal to EIDEA, the seam lies within one element. No checks are required.

- If EIDSA and EIDEA share two corner grids, these elements are adjacent. No checks are required.
- If EIDSA and EIDEA share only one corner grid, the seam is over a corner. There are two exceptions:

There exists a shell element (EIDMA) that shares two corner grids with EIDSA and EIDEA. Also, either the angle  $\theta$  between vector  $S_1S_2$  and vector  $E_1E_4$  is greater than CNRAGLI degrees or the middle point (M) of line segment  $S_2E_4$  projects to EIDSA, EIDMA, or EIDEA.

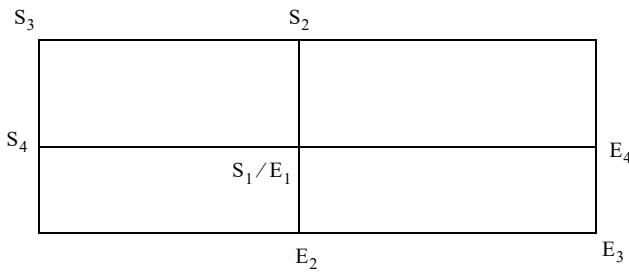


This model is acceptable - CONVEX ( $>$  CNRAGLI).



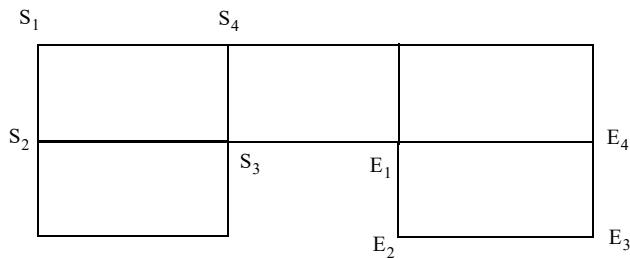
This model fails - not CONVEX ( $<$  CNRAGLI and point M does not project to EIDSA, EIDMA or EIDEA).

This shared grid is a shell grid of another two different shell elements.

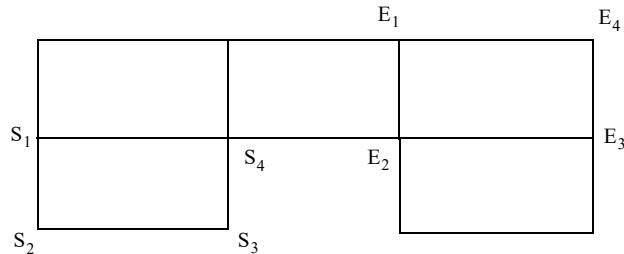


- If EIDSA and EIDEA do not share any corner grid, Nastran will check if there is an element (EIDMA) lying between EIDSA and EIDEA. EIDMA must share two corner grids with EIDSA and another different one corner grid with EIDEA, or vice versa. The following five examples demonstrate the acceptable and failed cases.

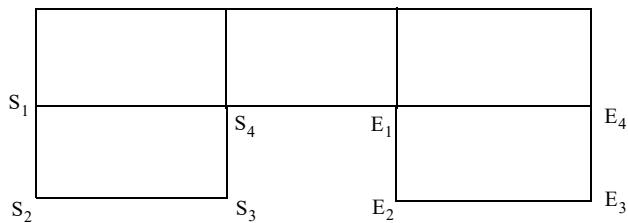
EIDMA shares one edge with EIDSA and shares one corner grid with EIDEA. This case is acceptable.



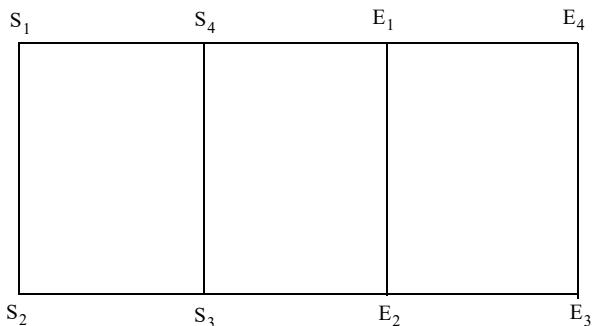
EIDMA shares one edge with IEDEA and shares one corner grid with EIDSA. This case is acceptable.



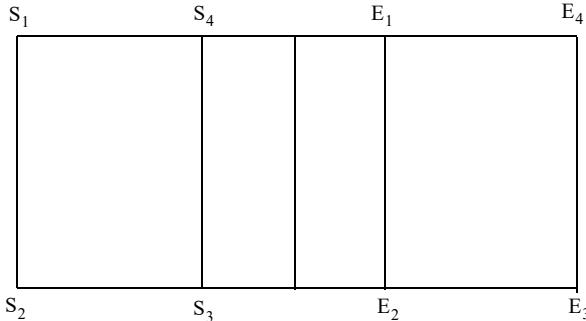
EIDMA shares one corner grid with EIDSA and shares another corner grid with EIDEA. An error is detected because the seam spans a cutout.



EIDMA shares one edge with EIDSA and shares another edge with EIDEA. This case is acceptable.

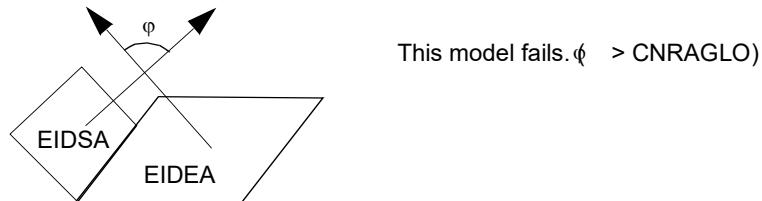


There does not exist a single element that shares an edge or corner grid with EIDSA or EIDEA. An error is detected because the length of the seam spans more than three elements.



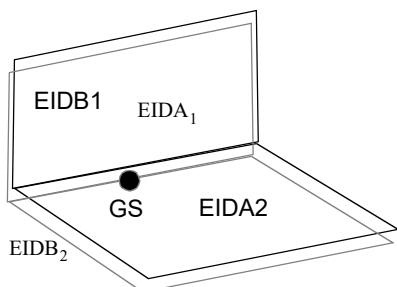
e. Check the CSEAM Over a Corner with Elements Out of Plane

The CNRAGLO parameter is used to check the error of a seam over a corner with EIDSA and EIDEA not lying on a same plane. An error is detected if the angle  $\varphi$  between the shell normal vectors of EIDSA and EIDEA is greater than CNRAGLO. The default value of CNRAGLO is  $20^\circ$ . No angles will be checked if CNRAGLO = -1.



f. Modeling Guidelines

When there exist multiple pairs of connections, it is recommended that either the GMCHK and GSPROJ flags be turned on to filter out tilted connections or the ELEM option be used to specify the IDs of the connected shell elements directly. For example, if EIDA1 is connected to EIDB2 or EIDA2 is connected to EIDB1, the element tangent vectors will be computed wrong and the auxiliary points will not be able to find connected shell elements.



6. The projection algorithm for the two-sided option will be the same as in Remark 5. above once the two patches A and B have been established. The program will find the closest shell grids to GS and GE as usual for candidate shell elements for patch A.

It will compute the normal for the candidate patch A (similar for GE) and for the candidate patch B. If the normals are approximately aligned (within a tolerance) the algorithm will proceed as in Remark 5.

If the normals of the candidate patch's A and B do not align within a specified tolerance, the algorithm will use another set of pairs of grids for candidate patches to find a new patch A and B. If their normals align within a specified tolerance it will proceed as in Remark 5. If after processing all reasonable pairs of patches, no alignment of normals are found or the patches A and B at GE have different normal alignment from the patches A and B at GB, a user fatal will be issued.

**Caution:** For the two-sided option, GS and GB must lie between patches A and B. Also, the shell elements that get projections from GS/GE cannot share a common shell grid. This option always selects the patch with the shell grids closest to GS/GE as patch A. Avoid having GS/GE exactly midway between the two patches.

7. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
8. The CSEAM contributes to MASS if its PSEAM entry has an associated MATi entry with a non-zero density. PARAM,COUPMASS effects the mass calculation. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
9. If partitioned superelements are present, then CSEAM is supported in the main Bulk Data section only.

**CSET****Free Boundary Degrees-of-Freedom**

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

| 1    | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|------|-----|----|-----|----|-----|----|-----|----|----|
| CSET | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|      |     |   |   |    |   |    |  |  |  |
|------|-----|---|---|----|---|----|--|--|--|
| CSET | 124 | 1 | 5 | 23 | 6 | 16 |  |  |  |
|------|-----|---|---|----|---|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.) |

**Remarks:**

1. CSET and BNDFREE entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**CSET1**

## Free Boundary Degrees-of-Freedom, Alternate Form of CSET Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

| 1     | 2   | 3   | 4    | 5      | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|------|--------|-----|-----|-----|-----|----|
| CSET1 | C   | ID1 | ID2  | ID3    | ID4 | ID5 | ID6 | ID7 |    |
|       | ID8 | ID9 | ID10 | -etc.- |     |     |     |     |    |

Example:

|       |     |   |   |   |   |   |    |     |  |
|-------|-----|---|---|---|---|---|----|-----|--|
| CSET1 | 124 | 1 | 5 | 7 | 6 | 9 | 12 | 122 |  |
|       | 127 |   |   |   |   |   |    |     |  |

Alternate Formats and Examples:

|       |   |     |        |     |  |  |  |  |  |
|-------|---|-----|--------|-----|--|--|--|--|--|
| CSET1 | C | ID1 | "THRU" | ID2 |  |  |  |  |  |
| CSET1 | 3 | 6   | THRU   | 32  |  |  |  |  |  |

|       |  |       |  |  |  |  |  |  |  |
|-------|--|-------|--|--|--|--|--|--|--|
| CSET1 |  | "ALL" |  |  |  |  |  |  |  |
| CSET1 |  | ALL   |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| C          | Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1 < ID2)  |

Remarks:

1. CSET1 and BNDFRE1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

## CSHEAR

## Shear Panel Element Connection

Defines a shear panel element and effective extensional stiffener rods.

Format:

| 1      | 2   | 3   | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|----|----|----|----|---|---|----|
| CSHEAR | EID | PID | G1 | G2 | G3 | G4 |   |   |    |

Example:

|        |   |   |   |   |   |   |  |  |  |
|--------|---|---|---|---|---|---|--|--|--|
| CSHEAR | 3 | 6 | 1 | 5 | 3 | 7 |  |  |  |
|--------|---|---|---|---|---|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)                                   |
| PID        | Property identification number of a PSHEAR entry. (Integer > 0; Default = EID)               |
| Gi         | Grid point identification numbers of connection points. (Integer > 0;<br>G1 ≠ G2 ≠ G3 ≠ G4 ) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.

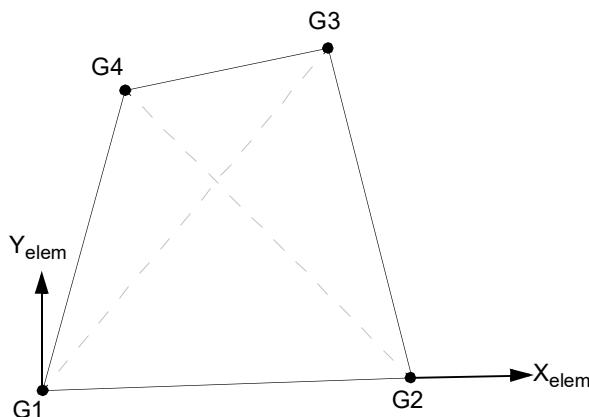


Figure 9-67 CSHEAR Element Connection and Coordinate System

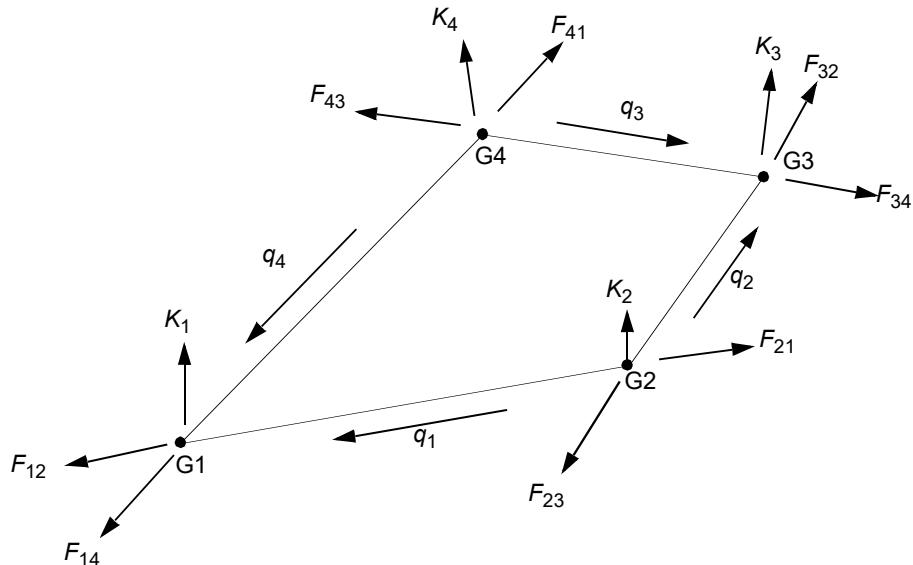
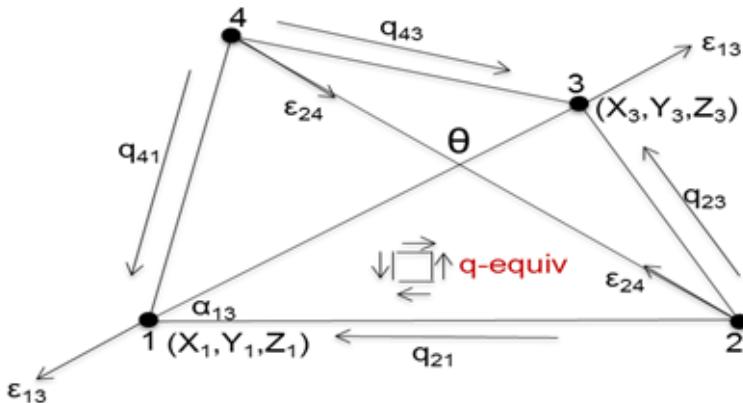


Figure 9-68 CSHEAR Element Corner Forces and Shear Flows

- The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: The Harder panel is based on the following:

Consider the following quadrilateral shear panel.



The shear stress  $\tau$  is related to the shear strain by the relationship  $\tau = G\gamma$  where  $G$  is the shear modulus.

Determining the value for shear for the general quadrilateral shown is a heuristic process. For the Harder element it is determined in the following manner. The strain along a diagonal such as L13 is found from the expression (or Mohr's circle):

$$\varepsilon_{13} = \varepsilon_x \cos^2 \alpha_{13} + \varepsilon_x \sin^2 \alpha_{13} + \gamma_{xy} \cos \alpha_{13} \sin \alpha_{13}$$

With a similar expression for  $\varepsilon_{24}$ , where  $\alpha_{24} = 180 - (\alpha_{13} + \theta)$ . For shear panels, the direct strains are negligible compared to the shear strain. For a rectangular shear panel  $\alpha_{24} = \alpha_{13}$  and we have

$$\Delta\varepsilon = \varepsilon_{13} - \varepsilon_{24} = \gamma \sin \theta \text{ or } \gamma = (\varepsilon_{13} - \varepsilon_{24}) / (\sin \theta).$$

In the above, the subscripts have been dropped from the term for shear strain. The Harder shear panel uses this definition for the measure of shear strain in the general quadrilateral. For a rectangular panel it is exact. For a reasonable panel it is within an error of < 1 degree. To insure reasonableness, all the geometry checks required by the Garvey element are still performed.

### Stress Recovery:

The output for either the Garvey or the Harder panel is the same with the following exception:

| MAX SHEAR                               | AVG SHEAR                                   | SHEARP |
|---|---|--------|
| MAX( $\tau_1, \tau_2, \tau_3, \tau_4$ ) | MAX( $\tau_1, \tau_2, \tau_3, \tau_4$ ) / 4 | GARVEY |
| MAX( $\tau_1, \tau_2, \tau_3, \tau_4$ ) | (q-Equiv) / t                               | HARDER |

For both Garvey and Harder panels, q-Equiv is the shear flow on which all stress calculations are based.

**CSLOT3****Three Point Slot Element Connection**

Defines an element connecting three points that solve the wave equation in two dimensions. Used in the acoustic cavity analysis for the definition of evenly spaced radial slots.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6 | 7   | 8 | 9 | 10 |
|--------|-----|------|------|------|---|-----|---|---|----|
| CSLOT3 | EID | IDS1 | IDS2 | IDS3 |   | RHO | B | M |    |

**Example:**

|        |     |   |   |   |  |       |  |   |  |
|--------|-----|---|---|---|--|-------|--|---|--|
| CSLOT3 | 100 | 1 | 3 | 2 |  | 3.0-3 |  | 6 |  |
|--------|-----|---|---|---|--|-------|--|---|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| IDSi       | Identification number of connected GRIDS points. (Integer > 0)   |
| RHO        | Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry)                      |
| B          | Fluid bulk modulus. (Real $\geq$ 0.0; Default is the value of BD on the AXSLOT entry)                            |
| M          | Number of slots in circumferential direction. (Integer $\geq$ 0; Default is the value of MD on the AXSLOT entry) |

**Remarks:**

1. CSLOT3 is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates three plot elements, connecting points IDS1 to IDS2, IDS2 to IDS3, and IDS3 to IDS1.
5. If B=0.0, then the slot is considered to be an incompressible fluid.
6. If M=0, then no matrices for CSLOT3 elements are generated.

**CSLOT4****Four Point Slot Element Connection**

Defines an element connecting four points that solve the wave equation in two dimensions. Used in acoustic cavity analysis for the definition of evenly spaced radial slots.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6    | 7   | 8 | 9 | 10 |
|--------|-----|------|------|------|------|-----|---|---|----|
| CSLOT4 | EID | IDS1 | IDS2 | IDS3 | IDS4 | RHO | B | M |    |

**Example:**

|        |     |   |   |   |   |  |       |   |  |
|--------|-----|---|---|---|---|--|-------|---|--|
| CSLOT4 | 101 | 1 | 3 | 2 | 4 |  | 6.2+4 | 3 |  |
|--------|-----|---|---|---|---|--|-------|---|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| IDSi       | Identification number of connected GRIDS points. (Integer > 0)  |
| RHO        | Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry.)                      |
| B          | Fluid bulk modulus. (Real $\geq$ 0.0; Default is the value of BD on the AXSLOT entry.)                            |
| M          | Number of slots in circumferential direction. (Integer $\geq$ 0; Default is the value of MD on the AXSLOT entry.) |

**Remarks:**

1. This entry is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates four plot elements connecting points IDS1 to IDS2, IDS2 to IDS3, IDS3 to IDS4, and IDS4 to IDS1.
5. If B = 0.0, then the slot is considered to be an incompressible fluid.
6. If M = 0, then no matrices for CSLOT4 elements are generated.

**CSPR****Springs With Offsets**

Springs for use in SOL 700 only.

Format:

| 1    | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|------|-----|-----|----|----|---|---|---|---|----|
| CSPR | EID | PID | G1 | G2 |   |   |   |   |    |

Example:

|      |   |    |     |     |  |  |  |  |  |
|------|---|----|-----|-----|--|--|--|--|--|
| CSPR | 1 | 22 | 456 | 457 |  |  |  |  |  |
|------|---|----|-----|-----|--|--|--|--|--|

| Descriptor | Meaning                                     | Type  | Default  |
|------------|---|-------|----------|
| EID        | Element ID. A unique number has to be used. | I > 0 | Required |
| PID        | Property ID of PSPRMAT entry                | I > 0 | Required |
| G1         | Gridpoint 1                                 | I > 0 | Required |
| G2         | Gridpoint 2.                                | I > 0 | Required |

**CSSCHD****Aerodynamic Control Surface Schedule Input**

Defines a scheduled control surface deflection as a function of Mach number and angle of attack.

**Format:**

| 1      | 2   | 3     | 4      | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|-----|-------|--------|-------|-------|---|---|---|----|
| CSSCHD | SID | AESID | LALPHA | LMACH | LSCHD |   |   |   |    |

**Example:**

|        |   |    |    |    |    |  |  |  |  |
|--------|---|----|----|----|----|--|--|--|--|
| CSSCHD | 5 | 50 | 12 | 15 | 25 |  |  |  |  |
|--------|---|----|----|----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)  |
| AESID      | ID of an AESURF Bulk Data entry to which the schedule is being attached.  |
| LALPHA     | ID of an AEFACT Bulk Data entry containing a list of angles of attack (in radians) at which schedule information is provided. (Integer > 0: Default = no angle information provided.) |
| LMACH      | ID of an AEFACT Bulk Data entry containing a list of Mach numbers at which schedule information is provided. (Integer > 0; Default = no Mach information provided)                    |
| LSCHD      | ID of an AEFACT Bulk Data entry which contains the scheduling information. See Remarks 4. and 5. (Integer > 0; no Default)  |

**Remarks:**

1. Control system schedules must be selected with the Case Control command CSSCHD = SID.
2. The AESID cannot appear on an AELINK or TRIM Bulk Data entry for the same subcase.
3. The control surface deflection is computed using a linear interpolation for the Mach number provided on the associated TRIM entry and the angle of attack derived as part of the trim calculation.
4. The LSCHD data are provided as a list of deflections (in radians) as a function of Mach numbers and angles of attack. If there are NMACH Mach numbers and NALPHA angles of attack, the first NALPHA deflections are for the first Mach number, the next NALPHA are for the second Mach number, and so on, until the last NALPHA deflections are for the final Mach number.
5. If LALPHA is blank, LSCHD contains NMACH deflections to define the Mach schedule. If LMACH is blank, LSCHD contains NALPHA deflections to define the angle of attack schedule.
6. LALPHA and LMACH cannot be simultaneously blank. If LALPHA or LMACH are not blank, at least two values of angle of attack or Mach number must be defined in order to perform interpolation.
7. If the Mach number or angle of attack is outside the range specified by the tabulated values, the value at the table end is used. That is, data are not extrapolated.

**CSSHL****Solid Shell Element Connection**

Defines a connection for a Solid Shell with 6 or 8 grid points in SOL 600 only.

**Format:**

| 1     | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|-----|-----|----|----|----|----|----|----|----|
| CSSHL | EID | PID | G1 | G2 | G3 | G4 | G5 | G6 |    |
|       | G7  | G8  |    |    |    |    |    |    |    |

**Example:**

|       |    |    |   |   |   |   |   |   |      |
|-------|----|----|---|---|---|---|---|---|------|
| CSSHL | 44 | 11 | 1 | 2 | 3 | 4 | 5 | 6 | quad |
|       | 7  | 8  |   |   |   |   |   |   |      |

|       |    |    |    |    |    |  |    |    |      |
|-------|----|----|----|----|----|--|----|----|------|
| CSSHL | 51 | 22 | 11 | 12 | 13 |  | 21 | 22 | tria |
|       | 23 |    |    |    |    |  |    |    |      |

|       |    |    |    |    |    |    |    |    |      |
|-------|----|----|----|----|----|----|----|----|------|
| CSSHL | 51 | 22 | 11 | 12 | 13 | 13 | 21 | 22 | tria |
|       | 23 | 23 |    |    |    |    |    |    |      |

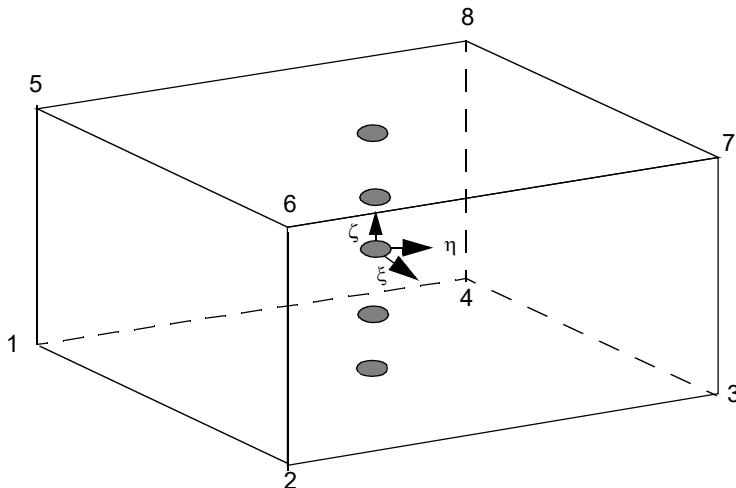
(Note: the 2nd and 3rd examples are equivalent to each other.)

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000; Required)  |
| PID        | Property identification of a PSSH entry. (Integer > 0; Required)  |
| Gi         | Grid point identification number of connection points. (Integer U or blank, for quad shapes all 8 values are required, for triangle shapes only G4 and G8 may be left blank in which case G4=G3 and G8=G7.) |

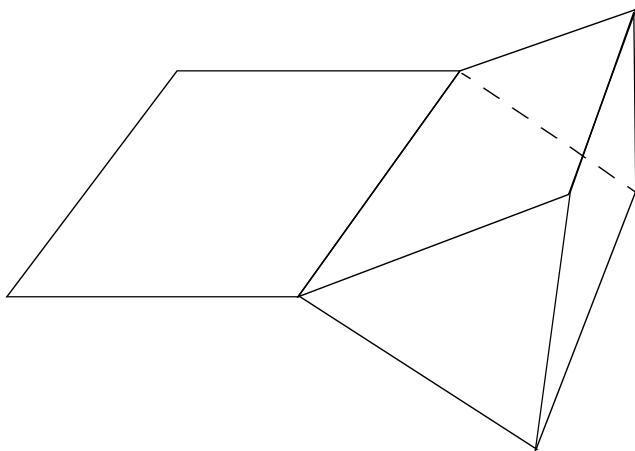
**Remarks:**

1. This element can degenerate to a triangle either by leaving G4 and G8 blank or by entering G4=G3 and G8=G7 (see 2<sup>nd</sup> and 3<sup>rd</sup> examples).
2. This element is usually only used when contact on each of the shell is anticipated.
3. Mid-side nodes are not available for this element.

4. Grid point ordering is shown in the following figure.



5. The stiffness of this element is formed using one integration point in the element plane and a user defined number through the element thickness. In this way the element can capture accurate material plasticity under bending load. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.
6. The number of integration points through the thickness is given by PARAM,MARCSLHT
7. This element may be collapsed to a triangular solid shell to attach to a standard shell such as CQUAD4 as follows:



8. This element is not currently available with Total Lagrange, finite strain plasticity or hyperelastic materials.

**CSSHLH****CHEXA to Solid Shell Element Connection**

Defines conversion of CHEXA elements to Solid Shell elements in SOL 600 only.

Format:

| 1      | 2    | 3   | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|--------|------|-----|------|------|------|------|------|---|----|
| CSSHLH | EID1 | PID | EID2 | EID3 | EID4 | EID5 | EID6 |   |    |

Example:

|        |    |    |    |     |     |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|
| CSSHLH | 44 | 11 | 54 | 200 | 250 |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EIDI       | CHEXA element identification numbers. See Remark 1. (0 < Integer < 100,000,000; no Default; EID1 is required) |
| PID        | Property identification of a PSSH entry to be used with all the elements with EIDI. (Integer > 0; Required)   |

1. All CHEXA elements in the range EID1 to EID2 will be converted to solid shells. The original CHEXA elements will be deleted and the solid shell elements will have the same element ID's as the original CHEXA elements.
2. All CHEXA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSSHL for additional items.
6. The PSSH entries associated with CSSHLH are not generated automatically. They must be input using a GUI or with a text editor.

**CSSHLM****CHEXA or CPENTA to Solid Shell Material Conversion**

Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements in SOL 600 only.

**Format:**

| 1      | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|------|------|------|------|------|------|------|------|----|
| CSSHLM | MID1 |      | MID2 | MID3 | MID4 | MID5 | MID6 | MID7 |    |
|        | MID8 | MID9 |      |      |      |      |      |      |    |

**Example:**

|        |     |  |     |     |       |  |  |  |  |
|--------|-----|--|-----|-----|-------|--|--|--|--|
| CSSHLM | 100 |  | 200 | 700 | 90250 |  |  |  |  |
|--------|-----|--|-----|-----|-------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MIDI       | Material identification numbers. See Remark 1. (Integer > 0; no Default; MID1 is required) |

**Remarks:**

1. All CHEXA and CPENTA elements with the MIDI specified will be converted to solid shells. The original CHEXA and CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original elements.
2. All PSOLID entries which reference all MIDI values will automatically be converted to PSSHL entries retaining the MID and CORDM values. The PSSHL property ID's will be the same as the PSOLID ID's plus IPOFF. The value of IPOFF should not be zero since the original PSOLID entries are not deleted and all property ID's should normally be unique.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSSHL for additional items.
6. If this entry is used, CSSHLH and CSSHLP entries should not be used.
7. See parameters MCSSHLCK and MCSSHORR for additional options for this entry.

**CSSHLH****CPENTA to Solid Shell Element Connection**

Defines conversion of CPENTA elements to Solid Shell elements in SOL 600 only.

**Format:**

| 1      | 2    | 3   | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|--------|------|-----|------|------|------|------|------|---|----|
| CSSHLH | EID1 | PID | EID2 | EID3 | EID4 | EID5 | EID6 |   |    |

**Example:**

|        |    |    |    |     |     |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|
| CSSHLH | 44 | 11 | 54 | 200 | 250 |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EIDi       | CPENTA element identification numbers. See Remark 1.<br>(0 < Integer < 100,000,000; no Default; EID1 is required) |
| PID        | Property identification of a PSSH entry to be used with all the elements with EIDi.<br>(Integer > 0; Required)    |

**Remarks:**

1. All CPENTA elements in the range EID1 to EID2 will be converted to solid shells. The original CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original CPENTA elements.
2. All CPENTA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
3. The solid shell elements will be collapsed Marc element 185 type elements with the 3<sup>rd</sup> and 4<sup>th</sup> grids set to G3 of the CPENTA and the 7<sup>th</sup> and 8<sup>th</sup> grids set to the 6<sup>th</sup> grid of the CPENTA.
4. Mid-side nodes are not allowed.
5. Please see the remarks for CSSHL for additional items.
6. The PSSH entries associated with CSSHLH are not generated automatically. They must be input using a GUI or with a text editor.

**CSUPER****Secondary Superelement Connection**

Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.

Format:

| 1      | 2    | 3    | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|------|------|--------|-----|-----|-----|-----|-----|----|
| CSUPER | SSID | PSID | GP1    | GP2 | GP3 | GP4 | GP5 | GP6 |    |
|        | GP7  | GP8  | -etc.- |     |     |     |     |     |    |

Example:

|        |        |    |   |   |   |    |  |  |  |
|--------|--------|----|---|---|---|----|--|--|--|
| CSUPER | 120003 | 21 | 3 | 6 | 4 | 10 |  |  |  |
|--------|--------|----|---|---|---|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SSID       | Coded identification number for secondary superelement. See Remark 1. (Integer > 0)   |
| PSID       | Identification number for referenced primary superelement. See Remark 2. (Integer > 0 or blank)                               |
| GPi        | Grid or scalar point identification numbers of the exterior points of the secondary superelement. See Remark 3. (Integer > 0) |

Remarks:

- The value of SSID is written in the form XXX0000 + n, where n is the referenced secondary superelement identification number and n must be less than 10000 and XXX is a displacement component sign reversal code as follows:  
The sign reversal code specifies the displacement component(s) normal to the plane of the mirror through which the reflection is to be made

Blank or 0      no reversal for identical superelement. If PSID is preceded by a minus sign and there is no xxx code on SSID, then a z-reversal mirror is generated.

|     |                     |   |                  |
|-----|---------------------|---|------------------|
| 1   | x-reversal          | } | Mirror Images    |
| 2   | y-reversal          |   |                  |
| 3   | z-reversal          |   |                  |
| 12  | x and y-reversal    | } | Identical Images |
| 23  | y and z-reversal    |   |                  |
| 31  | z and x-reversal    |   |                  |
| 123 | x, y and z-reversal | } | Mirror Images    |

2. If PSID = 0 or blank, the superelement boundary matrices are obtained from an external source (such as a database or external file). See also PARAM, [EXTDRUNT, 821](#).  
If PSID ≠ 0, the secondary superelement is identical to, or is a mirror image of, a primary superelement.
3. For identical or mirror image superelements, the grid point IDs, GPi, may appear in any order. However, if they are not in the same order as the external GRIDs of the primary superelement, then the SEQSEP entry is also required. In case of external superelements, the GRID IDs must be in the order that the terms in the associated matrices occur in.
4. Image superelements and their primaries must be congruent. The identical or mirror image superelement must have the same number of exterior grid points as its primary superelement. The exterior grid points of the image superelement must have the same relative location to each other as do the corresponding points of the primary superelement. The global coordinate directions of each exterior grid point of the image superelement must have the same relative alignment as those of the corresponding grid points of the primary superelement. If congruency is not satisfied because of round-off, then the tolerance may be adjusted with PARAM,CONFAC or DIAG 37.
5. For superelements from an external source, please refer to PARAMS [EXTDR, 821](#), [EXTDRUNT, 821](#) and [EXTUNIT, 824](#).

**CSUPEXT****Superelement Exterior Point Definition**

Assigns exterior points to a superelement.

**Format:**

| 1       | 2    | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|------|-----|-----|-----|-----|-----|-----|-----|----|
| CSUPEXT | SEID | GP1 | GP2 | GP3 | GP4 | GP5 | GP6 | GP7 |    |

**Example:**

|         |   |     |     |     |  |  |  |  |  |
|---------|---|-----|-----|-----|--|--|--|--|--|
| CSUPEXT | 2 | 147 | 562 | 937 |  |  |  |  |  |
|---------|---|-----|-----|-----|--|--|--|--|--|

**Alternate Format and Example:**

|         |      |     |        |     |  |  |  |  |  |
|---------|------|-----|--------|-----|--|--|--|--|--|
| CSUPEXT | SEID | GP1 | "THRU" | GP2 |  |  |  |  |  |
|---------|------|-----|--------|-----|--|--|--|--|--|

|         |   |       |      |       |  |  |  |  |  |
|---------|---|-------|------|-------|--|--|--|--|--|
| CSUPEXT | 5 | 12006 | THRU | 12050 |  |  |  |  |  |
|---------|---|-------|------|-------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Identification number of a primary superelement. (Integer > 0)   |
| GPi        | Grid or scalar point identification number in the downstream superelement or residual structure. (Integer > 0 or "THRU"; for "THRU" option, GP1 < GP2) |

**Remarks:**

1. Grid or scalar points are connected (that is, are exterior) to a superelement only if they are connected by structural, rigid, or plot elements. MPC entries are not regarded as elements. This entry is a means of providing connectivity for this purpose.
2. Open sets are allowed with the "THRU" option.
3. Scalar points may be interior to the residual structure (SEID = 0) only.
4. This entry may be applied only to the primary superelements. The CSUPER entry is used for secondary superelements (identical image, mirror image, and external superelements).

**CTETRA**

## Four-Sided Solid Element Connection

Defines the connections of the four-sided solid element with four to ten grid points.

**Format:**

| 1      | 2   | 3   | 4  | 5   | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|----|-----|----|----|----|----|----|
| CTETRA | EID | PID | G1 | G2  | G3 | G4 | G5 | G6 |    |
|        | G7  | G8  | G9 | G10 |    |    |    |    |    |

**Example:**

|        |     |    |   |    |    |   |     |     |  |
|--------|-----|----|---|----|----|---|-----|-----|--|
| CTETRA | 112 | 2  | 3 | 15 | 14 | 4 | 103 | 115 |  |
|        | 5   | 16 | 8 | 27 |    |   |     |     |  |

| Descriptor | Meaning  | Type                      | Default  |
|------------|--|---------------------------|----------|
| EID        | Element identification number.                               | 0 < Integer < 100,000,000 | Required |
| PID        | Property identification number of a PSOLID or PLSOLID entry. | Integer > 0               | Required |
| Gi         | Identification numbers of connected grid points.             | Integer $\geq 0$ or blank | Required |

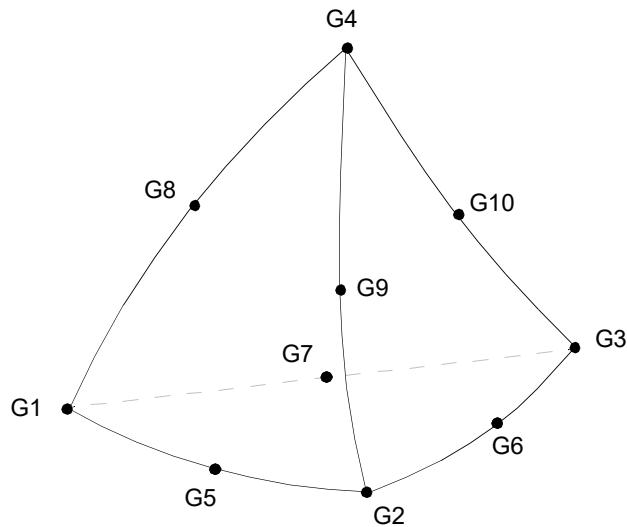


Figure 9-69      CTETRA Element Connection

### Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, i.e., G1, G2, G3 define a triangular face; G1, G8, and G4 are on the same edge, etc.
3. The edge points, G5 to G10, are optional. For Nastran conventional elements, any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system, except hyperelastic elements which outputs stress in the basic coordinate system.
5. For nonhyperelastic elements, the element coordinate system is derived from the three vectors R, S, and T, which join the midpoints of opposite edges.

R vector joins midpoints of edges G1-G2 and G3-G4.

S vector joins midpoints of edges G1-G3 and G2-G4.

T vector joins midpoints of edges G1-G4 and G2-G3.

The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that, if the R, S, and T vectors are described in the element coordinate system, a  $3 \times 3$  positive definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

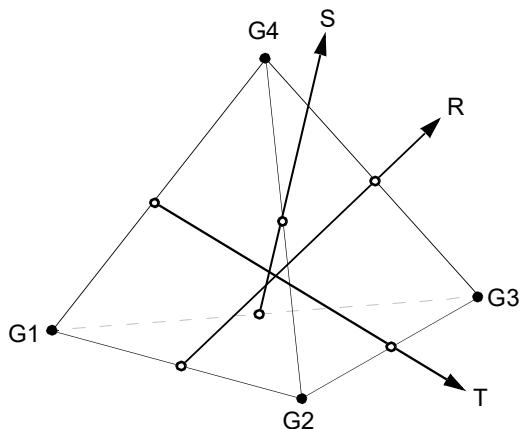


Figure 9-70 CTETRA Element R, S, and T Vectors

6. It is recommended that the edge points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CTETRAFD element name in [Item Codes, 1045](#).
8. If a CTETRA element is referenced by a PSET or PVAL entry, then p-version formulation is used and the element can have curved edges.
  - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
  - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the six edges of the element are considered straight unless:
  - For p-elements, there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
  - For h-elements, any of G5 through G10 are specified.
10. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-71](#).

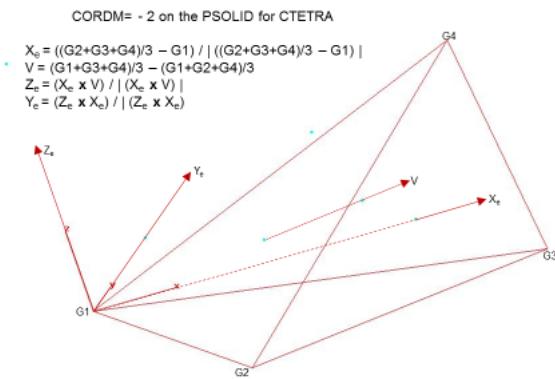


Figure 9-71 PSOLID on CTETRA

**CTRIA3****Triangular Plate Element Connection**

Defines an isoparametric membrane-bending or plane strain triangular plate element.

**Format:**

| 1      | 2   | 3     | 4  | 5  | 6  | 7             | 8     | 9 | 10 |
|--------|-----|-------|----|----|----|---------------|-------|---|----|
| CTRIA3 | EID | PID   | G1 | G2 | G3 | THETA or MCID | ZOFFS |   |    |
|        |     | TFLAG | T1 | T2 | T3 |               |       |   |    |

**Example:**

|        |     |     |      |      |      |     |      |  |  |
|--------|-----|-----|------|------|------|-----|------|--|--|
| CTRIA3 | 111 | 203 | 31   | 74   | 75   | 3.0 | 0.98 |  |  |
|        |     |     | 1.77 | 2.04 | 2.09 |     |      |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0; Default = EID)   |
| Gi         | Grid point identification numbers of connection points. (Integers > 0, all unique)   |
| THETA      | Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)  |
| MCID       | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R,      x -axis of MCID the coordinate is projected onto shell surface and the<br>CORD2R      material angle is measured from the G1-G2 line to the to the projected<br>x-axis<br><br>CORD1C,      r-axis of MCID the coordinate is projected onto shell surface through<br>CORD2C      the element center and the material angle is measured from the G1-G2<br>CORD1S,      line to the to the projected r-axis<br>CORD2S<br><br>Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer $\geq 0$ ; if blank, then THETA = 0.0 is assumed.) |

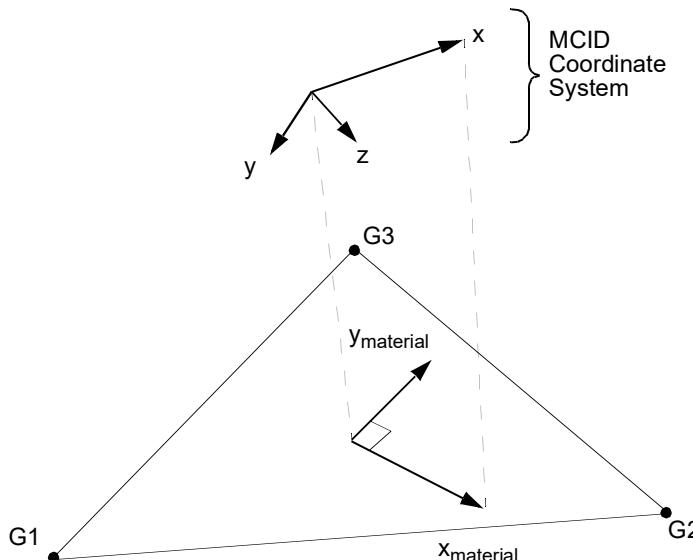


Figure 9-72 MCID Coordinate System Definition

|       |  |
|-------|--|
| ZOFFS | Offset from the surface of grid points to the element reference plane. See Remark 3.<br>ZOFFS is ignored for hyperelastic elements. (Real)   |
| TFLAG | An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)   |
| Ti    | Membrane thickness of element at grid points G1 through G3. If “TFLAG” zero or blank, then Ti are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the Ti are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank; not all zero. Default = 1.0) Ti are ignored for hyperelastic elements. |

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is highly recommended to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

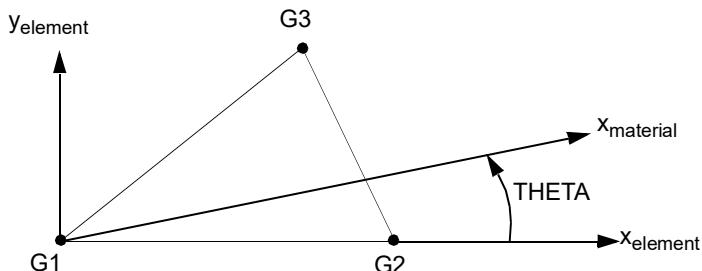


Figure 9-73 CTRIA3 Element Geometry and Coordinate Systems

4. The reference coordinate system for the output of stress, strain and element force depends on the element type.
  - For CTRIA3 elements, which are not p-elements and not hyperelastic, the reference coordinate system for output is the element coordinate system.
  - For CTRIA3 elements referenced by a PSET of PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system.
  - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
5. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes, 1045](#).
6. SYSTEM(218), alias T3SKEW, allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. The default value is 10. degrees.
7. If a CTRIA3 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
  - If a curved edge of a p-element is shared by an h-element CTRIA3, the geometry of the edge is ignored and set straight.

8. By default, all of the three edges of the element are considered straight unless the element is a p-element and the edges are associated to curved geometry with FEEDGE or FEFACE Bulk Data entries.
9. For RC network solver in thermal analysis, the ZOFFS is ignored.
10. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

**CTRIA6****Curved Triangular Shell Element Connection**

Defines a curved triangular shell element or plane strain with six grid points.

**Format:**

| 1      | 2                | 3     | 4  | 5  | 6  | 7     | 8  | 9  | 10 |
|--------|------------------|-------|----|----|----|-------|----|----|----|
| CTRIA6 | EID              | PID   | G1 | G2 | G3 | G4    | G5 | G6 |    |
|        | THETA<br>or MCID | ZOFFS | T1 | T2 | T3 | TFLAG |    |    |    |

**Example:**

|        |     |     |      |      |      |    |    |    |  |
|--------|-----|-----|------|------|------|----|----|----|--|
| CTRIA6 | 302 | 3   | 31   | 33   | 71   | 32 | 51 | 52 |  |
|        | 45  | .03 | .020 | .025 | .025 |    |    |    |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element Identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0)  |
| G1, G2, G3 | Identification numbers of connected corner grid points. (Unique Integers > 0)  |
| G4, G5, G6 | Identification number of connected edge grid points. Optional data for any or all three points for Nastran conventional element only. (Integer > 0 or blank)<br>Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.  |
| THETA      | Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)  |
| MCID       | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1R,    x -axis of MCID the coordinate is projected onto shell surface and the<br>CORD2R    material angle is measured from the G1-G2 line to the to the projected x-<br>axis<br><br>CORD1C,    r-axis of MCID the coordinate is projected onto shell surface through the<br>CORD2C    element center and the material angle is measured from the G1-G2 line<br>CORD1S,    to the to the projected r-axis<br>CORD2S<br><br>MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed.<br>(Integer $\geq$ 0; if blank, then THETA = 0.0 is assumed) |

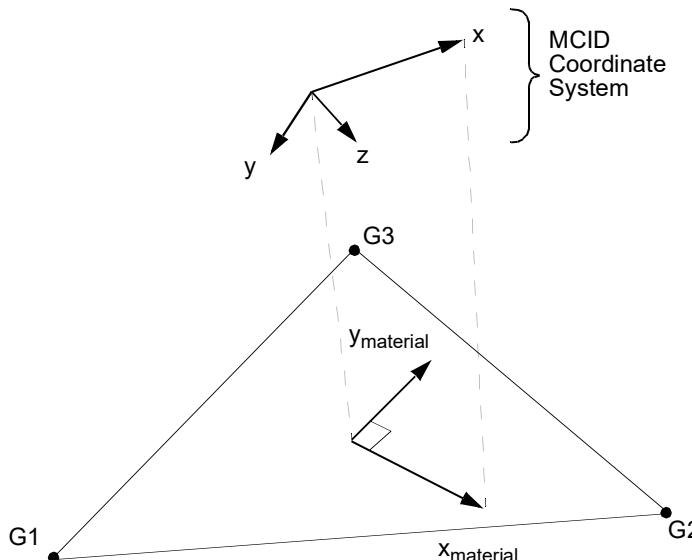


Figure 9-74 MCID Coordinate System Definition

|       |  |
|-------|--|
| ZOFFS | Offset from the surface of grid points to the element reference plane; see Remark 6.<br>ZOFFS is ignored for hyperelastic elements. (Real)   |
| Ti    | Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements. |
| TFLAG | An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)   |

#### Remarks:

1. Element identification numbers should be unique with respect to all other element IDs.
2. Grid points G1 through G6 must be numbered as shown in [Figure 9-75](#).
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between x<sub>material</sub> and the line of constant η.
4. T1, T2, and T3 are optional. If they are not supplied and no TFLAG, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge.

6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces will be incorrect. A User Warning Message is printed. A CTRIA3 element entry is recommended instead. If the element is hyperelastic, then the element is processed identically to the hyperelastic CTRIA3 element.
8. For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by  $x_l$  and  $y_l$  in [Figure 9-75](#). For hyperelastic elements, stresses and strains are output in the coordinate system defined by the CID field on the PLPLANE entry.

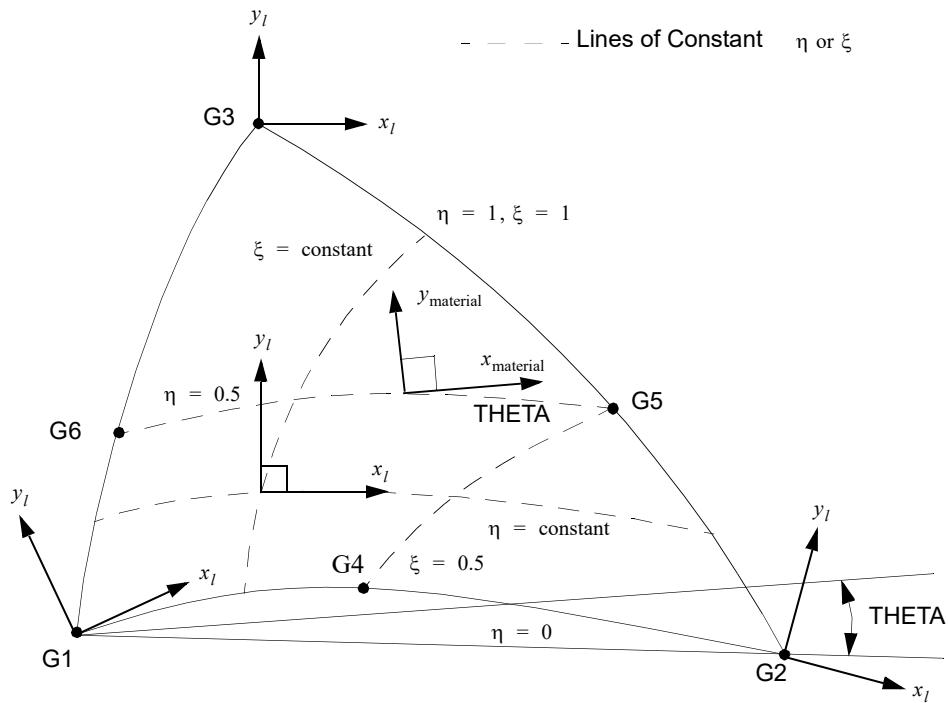


Figure 9-75 CTRIA6 Element Geometry and Coordinate Systems

9. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes, 1045](#).
10. For RC network solver in thermal analysis, the ZOFFS is ignored.
11. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

## CTRIAR

## Triangular Plate Element Connection

Defines an isoparametric membrane-bending triangular plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CQUADR element.

Format:

| 1      | 2   | 3     | 4  | 5  | 6  | 7             | 8     | 9 | 10 |
|--------|-----|-------|----|----|----|---------------|-------|---|----|
| CTRIAR | EID | PID   | G1 | G2 | G3 | THETA or MCID | ZOFFS |   |    |
|        |     | TFLAG | T1 | T2 | T3 |               |       |   |    |

Example:

|        |     |     |      |      |      |     |  |  |  |
|--------|-----|-----|------|------|------|-----|--|--|--|
| CTRIAR | 111 | 203 | 31   | 74   | 75   | 3.0 |  |  |  |
|        |     |     | 1.77 | 2.04 | 2.09 |     |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| PID        | Property identification number of a PSHELL, PCOMP or PCOMPG entry. (Integer > 0; Default = EID)  |
| G1, G2, G3 | Grid point identification numbers of connection points. (Integers > 0; all unique)   |
| THETA      | Material property orientation angle in degrees. (Real; Default = 0.0)  |
| MCID       | Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:<br><br>CORD1    x -axis of MCID the coordinate is projected onto shell surface and the R,    material angle is measured from the G1-G2 line to the to the projected x-<br>CORD2    axis<br>R<br><br>CORD1    r-axis of MCID the coordinate is projected onto shell surface through the C,    element center and the material angle is measured from the G1-G2 line to the to the projected r-axis<br>C<br>CORD1S<br>,<br>CORD2S |
|            | Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; if blank, then THETA = 0.0 is assumed)   |
| ZOFFS      | Offset from the surface of grid points to the element reference plane. See Remark 5.   |

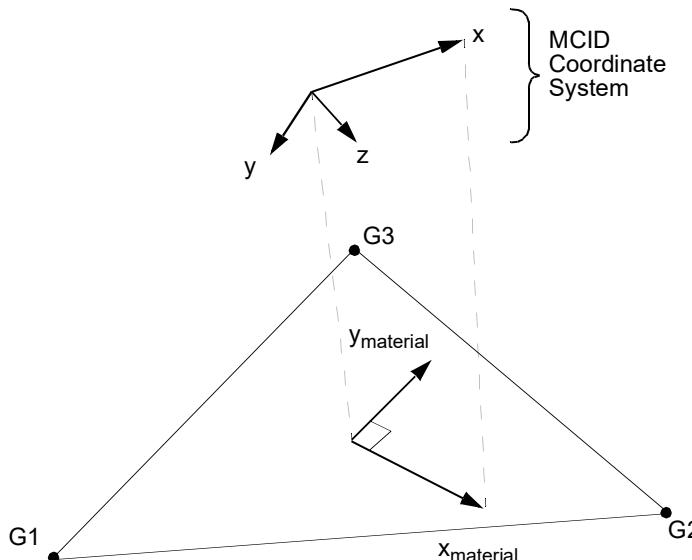


Figure 9-76 MCID Coordinate System Definition

TFLAG

An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Ti

Membrane thickness of element at grid points G1 through G4. If “TFLAG” zero or blank, then Ti are actual user specified thickness. (Real  $\geq 0.0$  or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the Ti are fraction relative to the T value of the PSHELL. (Real  $> 0.0$  or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Stresses are output in the element coordinate system at the centroid and grid points G1 through G3.
4. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4entry](#) (SORL option) and the [Consistent Surface and Edge Loads](#) (Ch. 3) in the *MSC Nastran Reference Guide* for additional information.

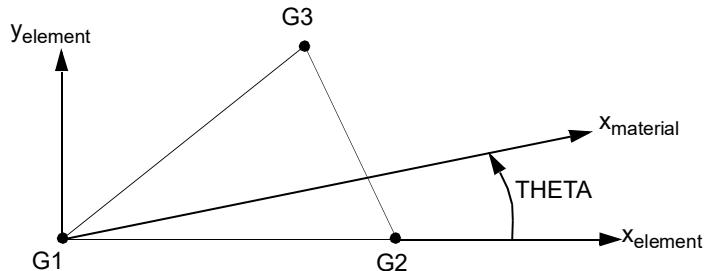


Figure 9-77 CTRIAR Element Geometry and Coordinate Systems

5. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

6. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

**CTRIAX****Axisymmetric Triangular Element (Fully Nonlinear or Linear Harmonic)**

Defines an axisymmetric triangular element with up to six grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis or a linear harmonic or rotordynamic analysis. The element has between three and six grid points.

**Format:**

| 1      | 2                | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|------------------|-----|----|----|----|----|----|----|----|
| CTRIAX | EID              | PID | G1 | G2 | G3 | G4 | G5 | G6 |    |
|        | THETA or<br>MCID |     |    |    |    |    |    |    |    |

**Example:**

|        |     |     |    |    |    |  |  |  |  |
|--------|-----|-----|----|----|----|--|--|--|--|
| CTRIAX | 111 | 203 | 31 | 74 | 75 |  |  |  |  |
|--------|-----|-----|----|----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000) See Remark 1.  |
| PID        | Property identification number of a PLPLANE or PAXSYMH entry. (Integer > 0). See Remark 2.  |
| G1, G2, G3 | Identification numbers of connected corner grid points. Required data for all three grid points. (Unique Integers > 0). See Remark 3., 6., 7.   |
| G4, G5, G6 | Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank). See Remark 3., 4., 6., 7.<br><br>Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified  |
| THETA      | Material property orientation angle in degrees. THETA is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Real; Default = 0.0)   |
| MCID       | Material coordinate system identification number, MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. The resulting angle is measured relative to the R axis of the element. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Integer $\geq 0$ ; if blank, then THETA = 0.0 is assumed.) THETA is ignored if PID refers to a PAXSYMH entry. |

**Remarks:**

1. Element identification numbers must be unique with respect to all other element IDs of any kind.

2. If PID refers to a PLPLANE entry, CTRIAX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CTRIAX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-78](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CTRIAXFD element name in [Item Codes, 1045](#).
6. The grid points of the axisymmetric element must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is divided by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50\text{N}$$

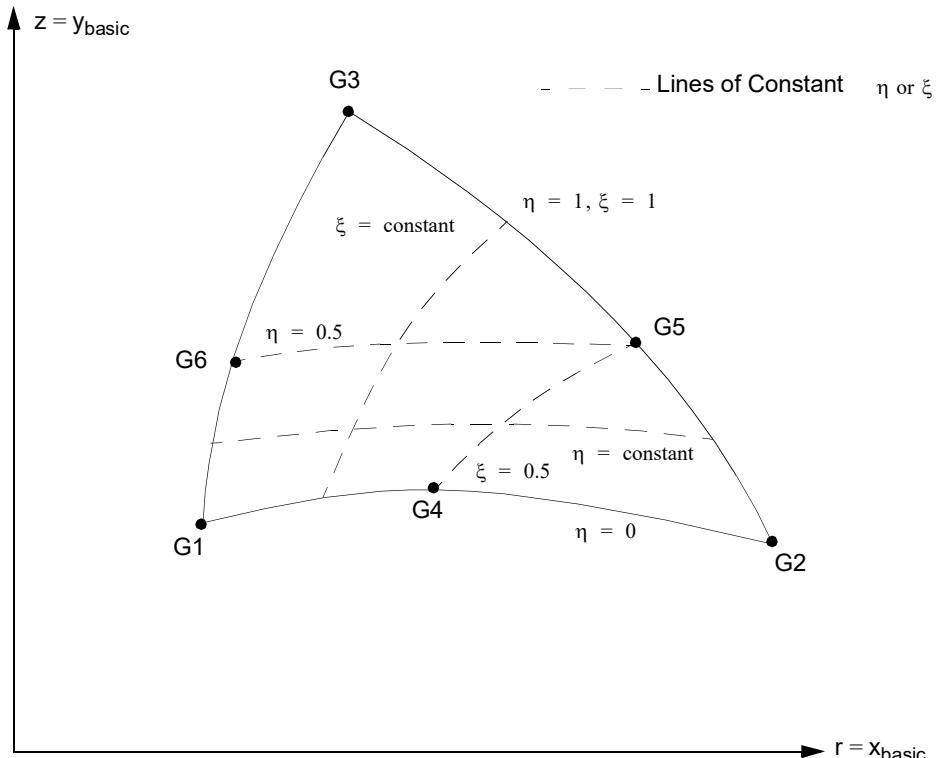


Figure 9-78 CTRIAX Element Coordinate System

**CTRIAX6****Axisymmetric Triangular Element Connection**

Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|-----|-----|----|----|----|----|----|----|----|
| CTRIAX6 | EID | MID | G1 | G2 | G3 | G4 | G5 | G6 |    |
|         | TH  |     |    |    |    |    |    |    |    |

**Example:**

|         |     |     |    |    |    |    |    |    |  |
|---------|-----|-----|----|----|----|----|----|----|--|
| CTRIAX6 | 22  | 999 | 10 | 11 | 12 | 21 | 22 | 32 |  |
|         | 9.0 |     |    |    |    |    |    |    |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| MID        | Material identification number. (Integer > 0)  |
| Gi         | Grid point identification numbers of connected points (unique Integers > 0, or blank for deleted nodes.) |
| TH         | Material property orientation angle in degrees. (Real; Default = 0.0)                                    |

**Remarks:**

1. The grid points must lie in the x-z plane of the basic coordinate system, with  $x = r \geq 0$ . The grid points must be listed consecutively beginning at a vertex and proceeding around the perimeter in either direction. Corner grid points G1, G3, and G5 must be present. Any or all edge grid points G2, G4, or G6 may be deleted for Nastran conventional element. Note that the alternate corner-edge grid point pattern is different from the convention used on the CTRIA6 element.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
2. For structural problems, the MID may refer to a MAT1 or MAT3 entry.
3. The continuation is optional.
4. Material properties (if defined on a MAT3 entry) and stresses are given in the  $(r_m, z_m)$  coordinate system shown in [Figure 9-80](#).
5. A concentrated load (e.g., FORCE entry) at Gi is multiplied by  $2\pi$  times the radius to Gi and then applied as a force per unit circumferential length. (which is located at a radius of 0.5 m), the magnitude of the load specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot 2\pi \cdot (0.5 \text{ m}) = 314.159 \text{ N}$$

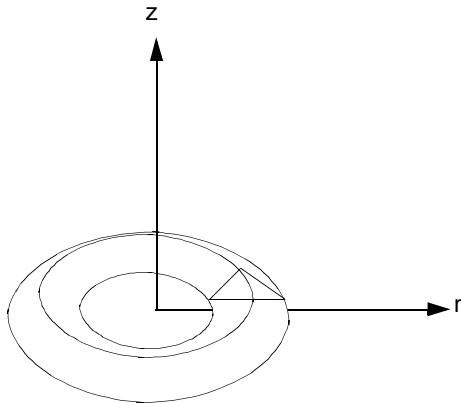


Figure 9-79 CTRIA6 Element Idealization

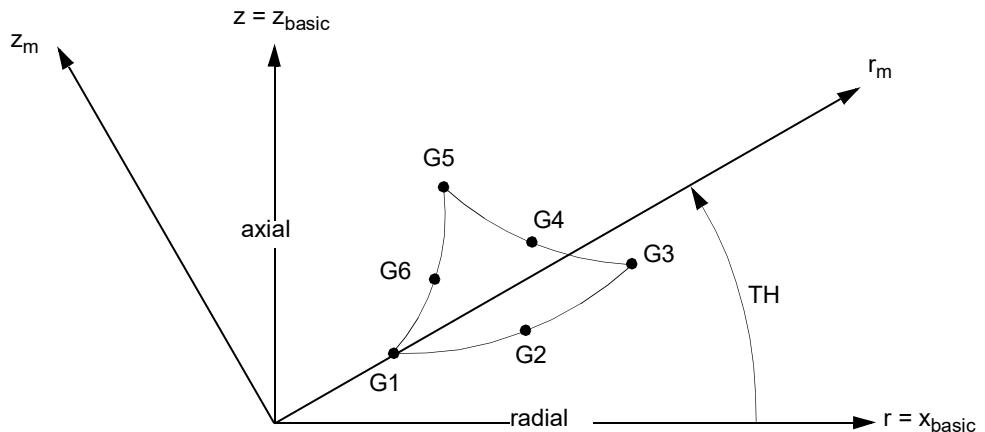


Figure 9-80 CTRIA6 Element Geometry and Coordinate Systems

6. For thermal problems, the MID may refer to a MAT4 or MAT5 entry. In order to model the convection or radiation along the edges the CHBDYG with TYPE=REV must be used.
7. To model axi-symmetric view factor computations using the CTRIA6 elements, the normal direction for the CTRIA6 element must point in the negative Y direction. The reason for this is to set up PATRAN for the correct nodal specification on the CHBDYG with REV option

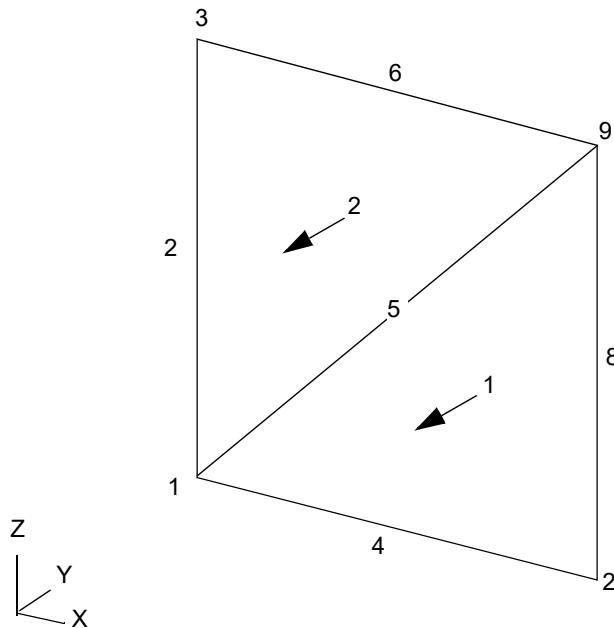


Figure 9-81      Normal Vectors That Point in a Negative Y Direction.

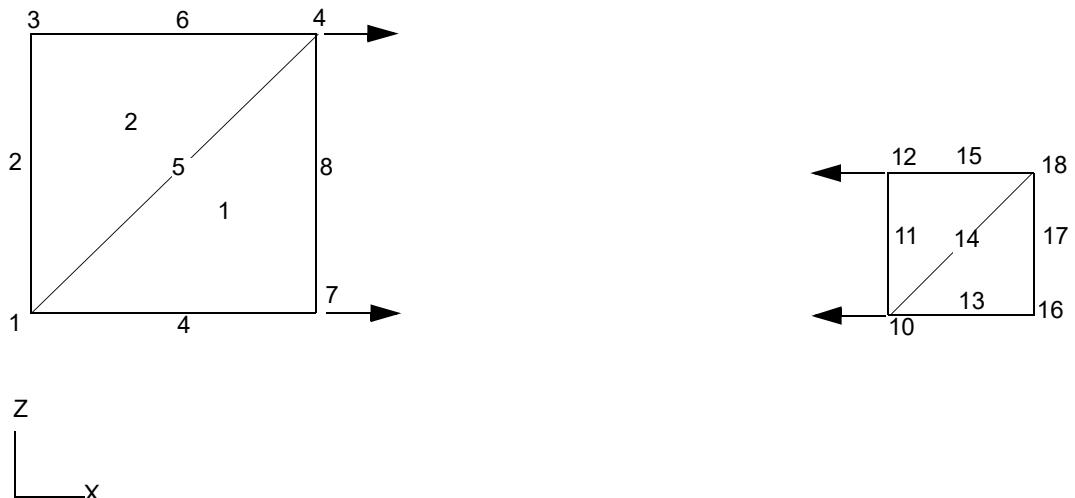


Figure 9-82      View Factor Boundary Conditions.

```
$ CHBDYG Surface Elements
CHBDYG 100001           REV      1
          7       9       8
CHBDYG 100002           REV      2
          12      10      11
```

Please note that on the CHBDYG, REV option, the nodal connectivity specification is in the clockwise fashion. In this case the CHBDYG with element ID of 100001 is defined by grids 7,9,8 which means the normal vector of the edge is pointing towards right. On the other hand the edges define by grid 12,10,11 means the normal vector of the edge is pointing towards left. If you have not reverse the element direction to a negative Y direction on the CTRIA6 element, the view factor computation will result a net view factor of zero.

**CTUBE**

## Tube Element Connection

Defines a tension-compression-torsion tube element.

Format:

| 1     | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|----|----|---|---|---|---|----|
| CTUBE | EID | PID | G1 | G2 |   |   |   |   |    |

Example:

|       |    |    |    |    |  |  |  |  |  |
|-------|----|----|----|----|--|--|--|--|--|
| CTUBE | 12 | 13 | 21 | 23 |  |  |  |  |  |
|-------|----|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)                     |
| PID        | Property identification number of a PTUBE entry. (Integer > 0; Default = EID)  |
| G1, G2     | Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one tube element may be defined on a single entry.

**CVISC****Viscous Damper Connection**

Defines a viscous damper element.

Format:

| 1     | 2   | 3   | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|----|----|---|---|---|---|----|
| CVISC | EID | PID | G1 | G2 |   |   |   |   |    |

Example:

|       |    |      |    |    |  |  |  |  |  |
|-------|----|------|----|----|--|--|--|--|--|
| CVISC | 21 | 6327 | 29 | 31 |  |  |  |  |  |
|-------|----|------|----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)                      |
| PID        | Property identification number of a PVISC entry. (Integer > 0; Default = EID)   |
| G1, G2     | Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ) |

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one viscous damper element may be defined on a single entry.
3. Grids G1 and G2 must not be coincident. If coincident grids are required, use either the CDAMP or CBUSH entry.

**CWELD****Weld or Fastener Element Connection**

Defines a weld or fastener connecting two surface patches or points. Large displacement and large rotational effects are supported when using SOL 600 and SOL 400 only.

**Format PARTPAT:**

| 1     | 2    | 3    | 4  | 5         | 6  | 7  | 8 | 9    | 10 |
|-------|------|------|----|-----------|----|----|---|------|----|
| CWELD | EWID | PWID | GS | "PARTPAT" | GA | GB |   | MCID |    |
|       | PIDA | PIDB |    |           |    |    |   |      |    |
|       | XS   | YS   | ZS |           |    |    |   |      |    |

**Example:**

|       |     |    |     |         |  |  |  |  |  |
|-------|-----|----|-----|---------|--|--|--|--|--|
| CWELD | 101 | 8  | 203 | PARTPAT |  |  |  |  |  |
|       | 21  | 33 |     |         |  |  |  |  |  |

**Alternate formats and examples:**

**Format ELPAT:**

|       |       |       |    |         |    |    |  |      |  |
|-------|-------|-------|----|---------|----|----|--|------|--|
| CWELD | EWID  | PWID  | GS | "ELPAT" | GA | GB |  | MCID |  |
|       | SHIDA | SHIDB |    |         |    |    |  |      |  |
|       | XS    | YS    | ZS |         |    |    |  |      |  |

**Example:**

|       |     |     |     |       |  |  |  |  |  |
|-------|-----|-----|-----|-------|--|--|--|--|--|
| CWELD | 103 | 5   | 403 | ELPAT |  |  |  |  |  |
|       | 309 | 511 |     |       |  |  |  |  |  |

**Format ELEMID:**

|       |       |       |    |          |    |    |  |      |  |
|-------|-------|-------|----|----------|----|----|--|------|--|
| CWELD | EWID  | PWID  | GS | "ELEMID" | GA | GB |  | MCID |  |
|       | SHIDA | SHIDB |    |          |    |    |  |      |  |

**Example:**

|       |     |     |     |        |  |  |  |  |  |
|-------|-----|-----|-----|--------|--|--|--|--|--|
| CWELD | 103 | 5   | 403 | ELEMID |  |  |  |  |  |
|       | 309 | 511 |     |        |  |  |  |  |  |

**Format GRIDID:**

|       |      |      |     |          |     |     |       |      |  |
|-------|------|------|-----|----------|-----|-----|-------|------|--|
| CWELD | EWID | PWID | GS  | "GRIDID" | GA  | GB  | SPTYP | MCID |  |
|       | GA1  | GA2  | GA3 | GA4      | GA5 | GA6 | GA7   | GA8  |  |
|       | GB1  | GB2  | GB3 | GB4      | GB5 | GB6 | GB7   | GB8  |  |

**Example:**

|       |    |    |     |        |    |    |    |    |  |
|-------|----|----|-----|--------|----|----|----|----|--|
| CWELD | 7  | 29 | 233 | GRIDID |    |    | QT |    |  |
|       | 15 | 28 | 31  | 35     | 46 | 51 | 55 | 60 |  |
|       | 3  | 5  | 8   |        |    |    |    |    |  |

**Format ALIGN:**

|       |      |      |  |         |    |    |  |      |  |
|-------|------|------|--|---------|----|----|--|------|--|
| CWELD | EWID | PWID |  | "ALIGN" | GA | GB |  | MCID |  |
|-------|------|------|--|---------|----|----|--|------|--|

**Example:**

|       |   |    |  |       |     |     |  |  |  |
|-------|---|----|--|-------|-----|-----|--|--|--|
| CWELD | 7 | 29 |  | ALIGN | 103 | 259 |  |  |  |
|-------|---|----|--|-------|-----|-----|--|--|--|

| Descriptor | Meaning   | Type                               | Default                                |
|------------|---|------------------------------------|--|
| EWID       | CWELD element identification number. See Remark 1.  | $0 < \text{Integer} < 100,000,000$ | Required                               |
| PWID       | Property identification number of a PWELD entry.  | Integer $> 0$                      | Required                               |
| GS         | Identification number of a grid point which defines the location of the connector. See Remarks 2. and 3.  | Integer $> 0$ or blank             |  |
| "PARTPAT"  | Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA and PIDB, respectively. The "PARTPAT" format connects up to 3x3 elements per patch. See Remark 4. | Character                          | Required                               |
| GA, GB     | Grid point identification numbers of piercing points on surface A and surface B, respectively. See Remark 5.  | Integer $> 0$ or blank             |  |
| MCID       | Specifies the element stiffness coordinate system. See Remark 16.   | Integer $\geq -1$ or blank         | Default = -1                           |
| PIDA, PIDB | Property identification numbers of PSHELL entries defining surface A and B respectively.  | Integer $> 0$                      | Required for "PARTPAT"                 |
| XS, YS, ZS | Coordinates of spot weld location in basic. See Remark 2.   | Real                               | Required if GS and GA are not defined. |

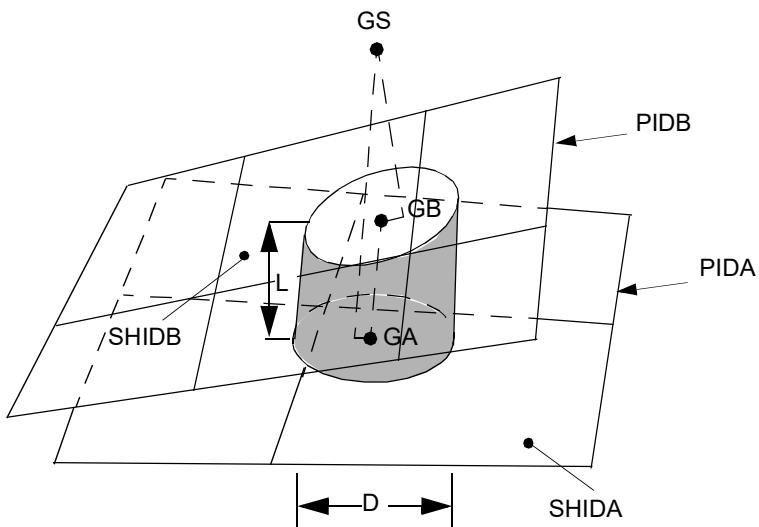
For the alternate formats, the descriptor meaning are described below:

| Descriptor      | Meaning   | Type        | Default               |
|-----------------|---|-------------|-----------------------|
| "ELPAT"         | Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELPAT" format connects up to 3x3 elements per patch. See Remark 6.      | Character   | Required              |
| SHIDA,<br>SHIDB | Shell element identification numbers of elements on patch A and B, respectively.  | Integer > 0 | Required for "ELPAT"  |
| "ELEMID"        | Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELEMID" format connects one shell element per patch. See Remark 7.      | Character   | Required              |
| SHIDA,<br>SHIDB | Shell element identification numbers of elements on patch A and B, respectively.  | Integer > 0 | Required for "ELEMID" |
| "GRIDID"        | Character string indicating that the connectivity of surface patch A to surface patch B is defined with two sequences of grid point identification numbers, GAi and GBi, respectively. The "GRIDID" format connects the surface of any element. See Remark 8. | Character   | Required              |
| SPTYP           | Character string indicating types of surface patches A and B. SPTYP = "QQ", "TT", "QT", "TQ", "Q" or "T". See Remark 9.   | Character   | Required for "GRIDID" |
| GAi             | Grid identification numbers of surface patch A. GA1 to GA3 are required. See Remark 10.   | Integer > 0 | Required for "GRIDID" |
| GBi             | Grid identification numbers of surface patch B. See Remark 10.  | Integer > 0 |                       |
| "ALIGN"         | Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA and GB, respectively. See Remark 11.  | Character   | Required              |
| GA, GB          | Vertex grid identification number of shell A and B, respectively.   | Integer > 0 | Required for "ALIGN"  |

#### Remarks:

1. CWELD defines a flexible connection between two surface patches, between a point and a surface patch, or between two shell vertex grid points. See [Figure 9-83](#) through [Figure 9-87](#).

2. Grid point GS defines the approximate location of the connector in space. GS is projected on surface patch A and on surface patch B. The resulting piercing points GA and GB define the axis of the connector. GS must have a normal projection on surface patch A and B. GS does not have to lie on the surface patches. GS is ignored for format "ALIGN". GA is used instead of GS if GS is not specified. For the formats "ELPAT" and "PARTPAT," if GS and GA are not specified, then XS, YS, and ZS will be assumed.
3. The connectivity between grid points on surface patch A and grid points on surface patch B is generated depending on the location of GS and the cross sectional area of the connector. Diagnostic print outs, checkout runs and non default settings of search and projection parameters are requested on the SWLDPRM Bulk Data entry. It is recommended to start with the default settings.
4. The format "PARTPAT" defines a connection of two shell element patches A and B with PSHELL property identification numbers PIDA and PIDB, respectively. The two property identification numbers must be different, see [Figure 9-83](#). The number of connected elements depends on the location of the piercing points GA, GB and the size of the diameter D. The diameter D is defined on the PWELD property entry. For this option, shell element patches A and B are allowed to share a common grid.



**Figure 9-83** Patch to Patch Connection Defined with the Formats PARTPAT or ELPAT

5. The definition of the piercing grid points GA and GB is optional for all formats with the exception of the format "ALIGN". If GA and GB are given, GS is ignored. If GA and GB are not specified, they are generated from the normal projection of GS on surface patches A and B and internal identification numbers are generated for GA and GB starting with 101e+6 by default. The offset number can be changed with PARAM,OSWPPT. If GA and GB are specified, they must lie on or at least have a projection on surface patches A and B, respectively. By default, the locations of user specified GA and

GB will not be changed. If the user specifies "SWLDPRM, MOVGAB, 1," then the locations will be corrected so that they lie on surface patches A and B within machine precision accuracy. The length of the connector is the distance of grid point GA to GB, subject to being adjusted to the effective length defined in PWELD entry.

6. The format "ELPAT" defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-83](#). The connectivity is similar to the format "PARTPAT". The number of connected elements depends on the location of the piercing points GA, GB and the size of the diameter D. For this option, shell element patches A and B are allowed to share a common grid.
7. The format "ELEMID" defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-84](#). The connectivity is restricted to a single element per patch regardless of the location of GA, GB and regardless of the size of the diameter of the connector. In addition, the format "ELEMID" can define a point to patch connection if SHIDB is left blank, see [Figure 9-85](#). Then grid GS is connected to shell SHIDA.

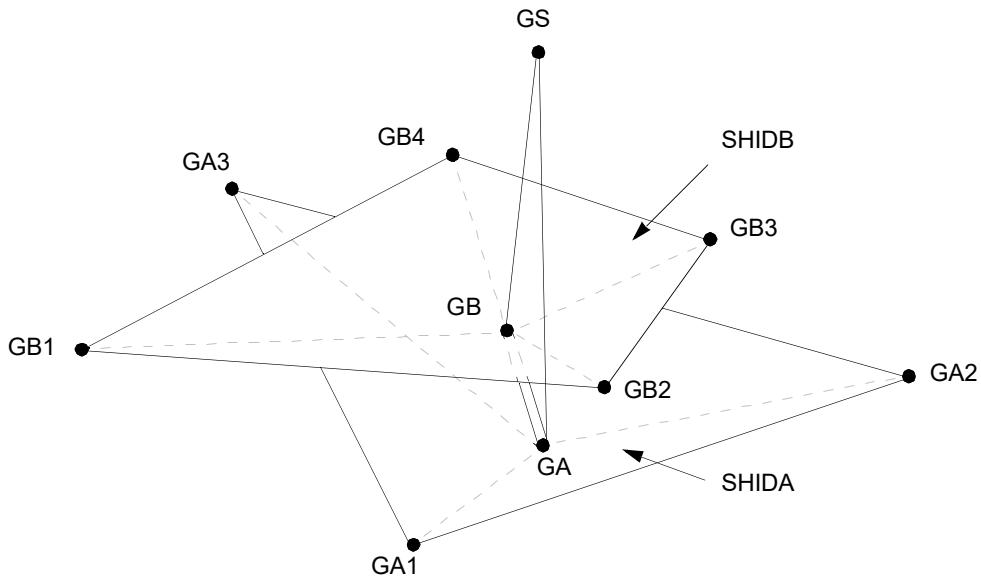


Figure 9-84 Patch to Patch Connection Defined with Format ELEMID or GRIDID

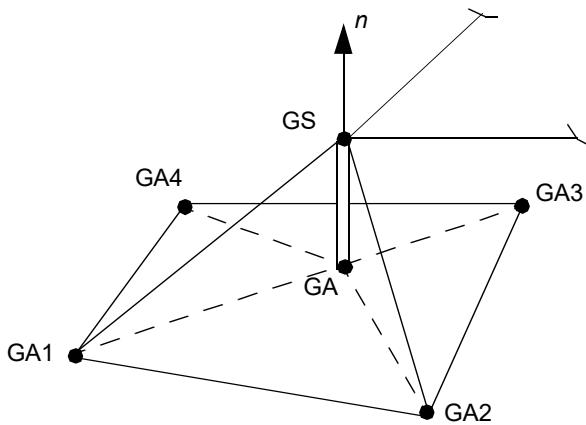
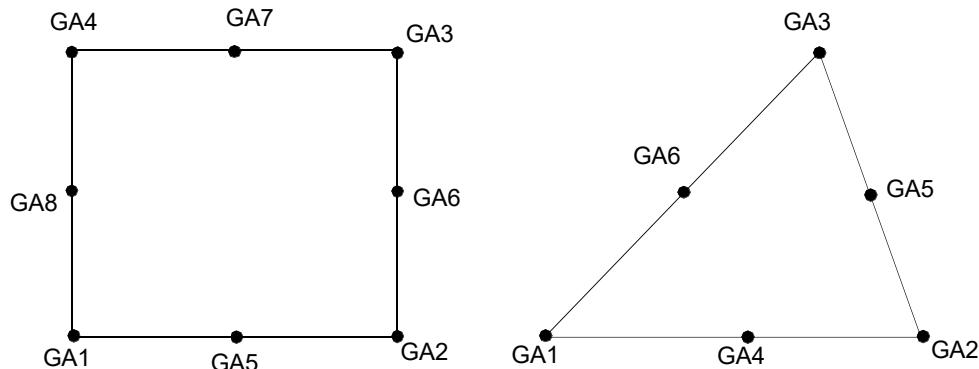


Figure 9-85 Point to Patch Connection Defined with Format ELEMID or GRID.

8. The format “GRIDID” defines a connection of two surface patches A and B with a sequence of grid points  $GA_i$  and  $GB_i$ , see [Figure 9-84](#). In addition, the format “GRIDID” can define a point to patch connection if all  $GB_i$  fields are left blank, see [Figure 9-85](#). Then grid GS is connected to grids  $GA_i$ . The grids  $GA_i$  and  $GB_i$  do not have to belong to shell elements.
9. SPTYP defines the type of surface patches to be connected. SPTYP is required for the format “GRIDID” to identify quadrilateral or triangular patches. The combinations are:

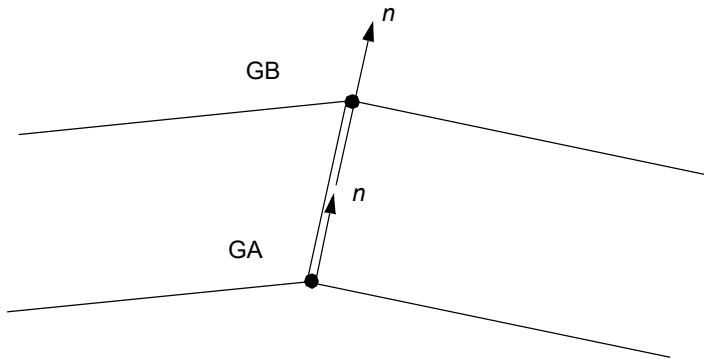
| SPTYP | Description  |
|-------|--|
| QQ    | Connects a quadrilateral surface patch A (Q4 to Q8) with a quadrilateral surface patch B (Q4 to Q8).                   |
| QT    | Connects a quadrilateral surface patch A (Q4 to Q8) with a triangular surface patch B (T3 to T6).                      |
| TT    | Connects a triangular surface patch A (T3 to T6) with a triangular surface patch B (T3 to T6).                         |
| TQ    | Connects a triangular surface patch A (T3 to T6) with a quadrilateral surface patch B (Q4 to Q8).                      |
| Q     | Connects the shell vertex grid GS with a quadrilateral surface patch A (Q4 to Q8) if surface patch B is not specified. |
| T     | Connects the shell vertex grid GS with a triangular surface patch A (T3 to T6) if surface patch B is not specified.    |

10. GAi are required for the format “GRIDID”. At least 3 and at most 8 grid IDs may be specified for GAi and GBi, respectively. The rules of the triangular and quadrilateral elements apply for the order of GAi and GBi, see [Figure 9-86](#). Missing midside nodes are allowed.



[Figure 9-86](#) Quadrilateral and Triangular Surface Patches defined with Format GRIDID

11. The format “ALIGN” defines a point to point connection, see [Figure 9-87](#). GA and GB are required, they must be existing vertex nodes of shell elements. For the other formats, GA and GB are not required. Two shell normals in the direction GA-GB are generated at GA and GB, respectively.



[Figure 9-87](#) Point to Point Connection Defined with Format ALIGN

12. Forces and moments are output in the element coordinate system, see [Figure 9-88](#). The element coordinate system is constructed using the following rules:

The element x-axis points from GA to GB.

$$e_1 = \frac{x_B - x_A}{\|x_B - x_A\|} \quad \text{element x-axis}$$

In case of zero length, the normal of shell A is taken. All vector components are in basic if not noted otherwise.

Find the smallest component  $j$  of  $e_1$

$$ce_1^i = |e_1^i|$$

$$ce_1^j = \min_{i=1,2,3} \{ce_1^i\}$$

Note that  $ce_1^i$  will be set to  $10^{-6}$  if  $ce_1^i < 10^{-6}$ .

In case of two equal components we take the one with the smaller  $i$ . The corresponding basic vector

$$b_j, \text{ e.g., for } j=3, b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

provides a good directional choice for  $e_2$ . In addition, the vector  $e_2$  must be orthogonal to  $e_1$ .

$$\tilde{e}_2 = b_j - \frac{e_1^T b_j}{e_1^T e_1} e_1 \quad e_2 = \frac{\tilde{e}_2}{\|\tilde{e}_2\|} \text{ element y-axis}$$

and  $e_3$  is just the cross product

$$e_3 = e_1 \times e_2 \quad \text{element z-axis}$$

The final transformation matrix is

$$T_{be} = \left[ \begin{array}{c|c|c} & e_1 & e_2 & e_3 \\ \hline e_1 & & & \end{array} \right]$$

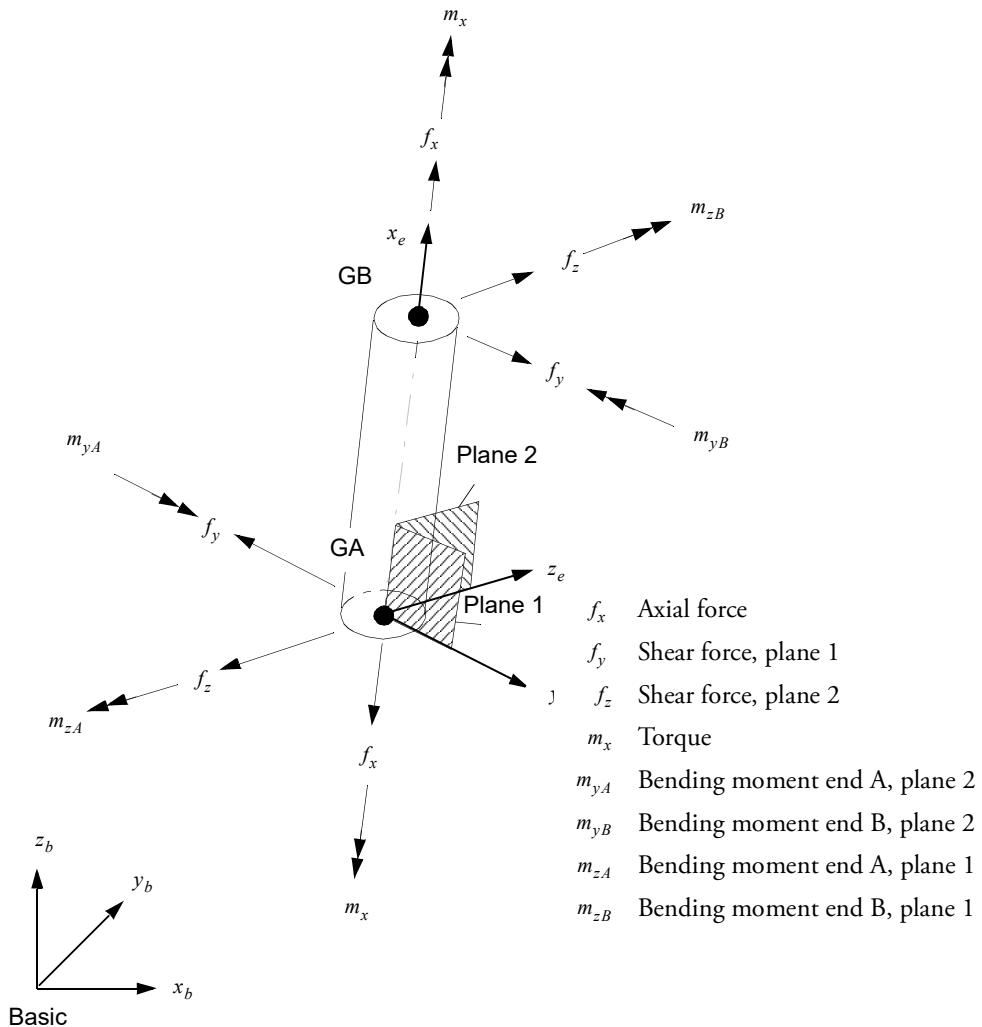


Figure 9-88 Element Coordinate System and Sign Convention of Element Forces

13. If "PARAM, OLDWELD, YES", in a SOL 400 analysis the CWELD element using method ELEMID, GRIDID, ELPAT or PARTPAT internally gets decomposed in a CBEAM element and a number of RBE3 elements. For method ALIGN the CWELD element internally gets replaced by a CBEAM element. The CBEAM element obtains a circular cross section with its diameter and material properties taken from the PWELD input and its length is determined by the final locations of the GA and GB grids of the CWELD. The RBE3 elements connect the GA and GB grids to the plate structures. For methods ELEMID and GRIDID one RBE3 element is used on each side to establish the connection. For methods ELPAT and PARTPAT five RBE3 elements are used on each side to establish the connection.

14. The output format of the forces and moments including the sign convention is identical to the CBAR element, see [Element Force Item Codes, 1085](#).
15. This entry is not supported in SOL 700.
16. MCID = -1 or blank (Default), then the coordinate system is as defined in Remark [12](#).  
If MCID  $\geq 0$ , then a “beam” like coordinate system is defined. The  $x_{elem}$  axis direction of the connector defined as

$$\hat{e}_1 = \frac{\hat{x}_B - \hat{x}_A}{\|\hat{x}_B - \hat{x}_A\|}$$

The T2 direction defined by MCID will be used to define the orientation vector  $\vec{v}$  of the connector. Then the  $z_{elem}$  element axis will be defined as

$$\hat{e}_3 = \frac{\hat{e}_1 \times \vec{v}}{\|\hat{e}_1 \times \vec{v}\|}$$

The element  $y_{elem}$  axis is defined as

$$\hat{e}_2 = \hat{e}_3 \times \hat{e}_1$$

17. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
18. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
19. If partitioned superelements are present, then CWELD is supported in the main Bulk Data section only.

**CYAX**

## Grid Points on Axis of Symmetry

Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.

**Format:**

| 1    | 2  | 3   | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|------|----|-----|--------|----|----|----|----|----|----|
| CYAX | G1 | G2  | G3     | G4 | G5 | G6 | G7 | G8 |    |
|      | G9 | G10 | -etc.- |    |    |    |    |    |    |

**Example:**

|      |    |     |      |     |     |    |  |  |  |
|------|----|-----|------|-----|-----|----|--|--|--|
| CYAX | 27 | 152 | THRU | 160 | 192 | 11 |  |  |  |
|------|----|-----|------|-----|-----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| Gi         | A list of grid points on the axis of symmetry. (Integer > 0 or Character “THRU”) |

**Remarks:**

1. The displacement coordinate system (see CD field on GRID entry) for a grid point lying on the axis of symmetry must be a rectangular system with the z-component of motion aligned with the axis of symmetry. The positive axis of symmetry is defined so that the azimuthal angle from positive side 1 to side 2 of a segment is in the same direction as the angle from T1 to T2 for the axis point. This is consistent with left- or right-hand rule.
2. If the dihedral symmetry option (STYPE = “DIH” on the CYSYM entry) is selected, the y-axis must be perpendicular to side 1.
3. Grid points lying on the axis of symmetry may be constrained by SPCs but not by MPCs. If the number of segments is three or more, SPCs must be applied to both components 1 and 2 or to neither, and SPCs must be applied to both components 4 and 5 or to neither in order to satisfy symmetry. In addition, the degrees-of-freedom (not constrained by SPCs, if any) at these grid points must be in the analysis set (a-set). If all degrees-of-freedom of grid points on the axis of symmetry are constrained by SPCs (including heat transfer, where there is only one degree-of-freedom), the grid point should not be listed on the CYAX entry.
4. Grid points lying on the axis of symmetry must not be defined on side 1 or on side 2 by means of a CYJOIN entry.
5. The word “THRU” must not appear in fields 2 or 9.

**CYJOIN****Cyclic Symmetry Boundary Points**

Defines the boundary points of a segment in cyclic symmetry problems.

**Format:**

| 1      | 2    | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|--------|------|----|----|--------|----|----|----|----|----|
| CYJOIN | SIDE | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|        | G7   | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|        |    |      |    |   |    |      |    |    |  |
|--------|----|------|----|---|----|------|----|----|--|
| CYJOIN | 1  | T2   | 7  | 9 | 16 | THRU | 33 | 64 |  |
|        | 72 | THRU | 89 |   |    |      |    |    |  |

| Descriptor | Meaning   |
|------------|---|
| SIDE       | Side identification. (Integer 1 or 2)   |
| C          | Type of coordinate system used on boundaries of dihedral or axisymmetry problems.<br>See Remark 3. (Character: "T1", "T2", "T3", "R", "C", "S") |
| Gi         | Grid or scalar point identification numbers. (Integer > 0 or Character "THRU")  |

**Remarks:**

1. CYJOIN entries are used only for cyclic symmetry problems. The CYSYM entry must be used to specify rotational, dihedral, or axisymmetry.
2. For rotational or axisymmetry problems, there must be one logical entry for SIDE = 1 and one for SIDE = 2. The two lists specify grid points to be connected; therefore, both lists must have the same length.
3. For dihedral problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. For dihedral and/or AXI type of symmetry, the grid point degree-of-freedom that is normal to the boundary must be specified in field 3 as "T1", "T2", or "T3" ("R", rectangular, and "C", cylindrical, are the same as "T2" while "S", spherical, is the same as "T3"). For scalar and extra points with one degree-of-freedom, these should be specified as blank, "T2", or "T3" if they are to have the same sign, and "T1", if the two connected points are to be opposite in sign.
4. All components of displacement at boundary points are connected to adjacent segments except those constrained by SPCi, MPC, or OMITi entries.
5. The points on the axis of symmetry of the model, defined in the CYAX entry must not be defined as a side 1 or side 2 point by means of this entry.
6. The word "THRU" may not appear in fields 4 or 9 of the parent entry and fields 2 or 9 on the continuation entries.
7. All grid points that are implicitly or explicitly referenced must be defined.

8. For rotational and axisymmetry problems, the displacement coordinate systems must be consistent between sides 1 and 2. This is best satisfied by the use of a spherical or cylindrical coordinate system.

**CYLINDR**

Defines the Shape of a Cylinder

Cylindrical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3 | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|-----|---|-----|-----|-----|-----|-----|-----|----|
| CYLINDR | VID |   | XC1 | YC1 | ZC1 | XC2 | YC2 | ZC2 |    |
|         | RAD |   |     |     |     |     |     |     |    |

**Example:**

|         |    |  |    |    |    |    |    |    |  |
|---------|----|--|----|----|----|----|----|----|--|
| CYLINDR | 4  |  | 0. | 0. | 0. | 1. | 1. | 1. |  |
|         | .5 |  |    |    |    |    |    |    |  |

| Descriptor    | Meaning  |
|---------------|--|
| VID           | Unique cylinder number. (Integer > 0; Required)        |
| XC1, YC1, ZC1 | Coordinates of point 1, See Remark 1. (Real; Required) |
| XC2, YC2, ZC2 | Coordinates of point 2. See Remark 1. (Real; Required) |
| RAD           | Radius of the cylinder. (Real; Required)               |

**Remarks:**

1. A cylinder is defined by the two end points of the cylinder axis and a radius.
2. Initial conditions are defined for the elements that are fully or partially inside the cylinder.
3. See also the TICEUL1 Bulk Data entry.

**CYSUP**

## Fictitious Supports for Cyclic Symmetry

Defines fictitious supports for cyclic symmetry analysis.

**Format:**

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|---|---|---|---|---|---|----|
| CYSUP | GID | C |   |   |   |   |   |   |    |

**Example:**

|       |    |      |  |  |  |  |  |  |  |
|-------|----|------|--|--|--|--|--|--|--|
| CYSUP | 16 | 1245 |  |  |  |  |  |  |  |
|-------|----|------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| GID        | Grid point identification number. (Integer > 0)  |
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks.) |

**Remarks:**

- Components of motion defined on this entry may not appear on SPC, SPC1, OMIT, OMIT1 entries, or in rigid elements or multipoint constraints as dependent degrees-of-freedom.
- Supports are applied at the grid point identified in field 2 to prevent rigid body motions in static analysis, or to define rigid body modes in eigenvalue analysis. All degrees-of-freedom should be at a single grid point. In other words, there can only be one such supported grid point in the model. The supports are applied only to the cyclic components of order k=0 or k=1. In order to satisfy conditions of symmetry, certain restrictions are placed on the location of the grid point and the orientation of its displacement coordinate system, as shown in the following table:

|  |   |                             |        |                             |                             |
|--|---|-----------------------------|--------|-----------------------------|-----------------------------|
| Symmetry Option (STYPE on the CYSYM entry)             | ROT   | ROT                         | DIH    | DIH                         | DIH                         |
| Number of Segments, N                                  | 2   | $\geq 3$                    | 1      | 2                           | $\geq 3$                    |
| Location of Grid Point                                 | See Note <a href="#">c.</a>                         | See Note <a href="#">d.</a> | Side 1 | Side 1                      | Side 1                      |
| Special Restrictions on Displacement Coordinate System | See Notes <a href="#">a.</a> and <a href="#">e.</a> | See Note <a href="#">b.</a> | None   | See Note <a href="#">a.</a> | See Note <a href="#">b.</a> |

**Notes:**

- T3 axis must be parallel to axis of symmetry.
- Displacement coordinate system at the referenced grid point must be cylindrical with z-axis along the axis of symmetry.

- c. Any location except on side 2.
  - d. Any location except on the axis of symmetry or on side 2.
  - e. If the grid point is on the axis of symmetry, the displacement coordinate system must be rectangular.
3. If the number of segments, N, is 1 (in the case of DIH symmetry) or 2 (in the case of ROT or AXI symmetry), it is important that the rotational components referenced in field 3 be elastically connected to the structure. If  $N \geq 2$  (in the case of DIH symmetry) or  $N \geq 3$  (in the case of ROT or AXI symmetry), it is not important, because in this case the supports for rigid body rotation are actually applied to translational motions.
  4. If  $N \geq 3$ , supports will be applied to both the 1 and 2 (inplane-translational) components, if either is referenced, and to both the 4 and 5 (out-of-plane rotational) components, if either is referenced. If component 6 is supported, component 2 should not appear on OMIT or OMIT1 entries.
  5. The restrictions noted in Remarks 2. and 4. are related to symmetry requirements. For  $N \geq 3$ , symmetry requires that the supports be symmetrical (or antisymmetrical), with respect to any plane passing through the axis of symmetry. For the DIH options,  $N = 1$  and  $N = 2$ , symmetry requires that the supports be symmetrical (or antisymmetrical) with respect to the plane(s) of symmetry. For the ROT option,  $N = 2$ , symmetry requires that a support be either parallel or perpendicular to the axis of symmetry.
  6. GID must be a grid point and not a scalar point.

**CYSYM****Cyclic Symmetry Parameters**

Defines parameters for cyclic symmetry analysis.

Format:

| 1     | 2    | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|------|-------|---|---|---|---|---|---|----|
| CYSYM | NSEG | STYPE |   |   |   |   |   |   |    |

Example:

|       |   |     |  |  |  |  |  |  |  |
|-------|---|-----|--|--|--|--|--|--|--|
| CYSYM | 6 | ROT |  |  |  |  |  |  |  |
|-------|---|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NSEG       | Number of segments. (Integer > 0)                  |
| STYPE      | Symmetry type. (Character: "ROT", "DIH", or "AXI") |

Remarks:

1. STYPE = "AXI" is a special case of STYPE = "ROT" used to model axisymmetric structures.
2. If STYPE = "AXI", then all grid points must lie on side 1, side 2, or the axis. Also, plate elements with midside grid points may not be defined.

## +Entries D - E

### DAMPING

### Parameter and Hybrid Damping Specification

Specifies the values for parameter damping and/or selects optional HYBRID damping.

Format:

| 1       | 2  | 3  | 4      | 5      | 6      | 7      | 8      | 9 | 10 |
|---------|----|----|--------|--------|--------|--------|--------|---|----|
| DAMPING | ID | G  | ALPHA1 | ALPHA2 | HYBRID | GEFACT | ROTSEP |   |    |
|         | W3 | W4 | WH     |        |        |        |        |   |    |

Example:

|         |     |     |  |  |  |  |    |  |  |
|---------|-----|-----|--|--|--|--|----|--|--|
| DAMPING | 100 | .02 |  |  |  |  | NO |  |  |
|---------|-----|-----|--|--|--|--|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Damping entry identification number. (Integer > 0; no Default)   |
| G          | Structural damping coefficient, see Remark 1. (Real; Default = 0.0)  |
| ALPHA1     | Scale factor for mass portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)   |
| ALPHA2     | Scale factor for stiffness portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)  |
| HYBRID     | Identification number of HYBDAMP entry for hybrid damping, see Remarks 5. and 6. (Integer $\geq$ 0; Default = 0)                       |
| GEFACT     | Scale factor for material damping. See Remark 1. and Remark 3. (Real; Default = 1.0)   |
| ROTSEP     | Remove rotor stiffness, mass, and structural damping from the hybrid damping calculation (Character: YES or NO; Default=YES)           |
| W3         | Average frequency for calculation of structural damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0)          |
| W4         | Average frequency for calculation of material damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0)            |
| WH         | Average frequency for calculation of hybrid ‘structural’ damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0) |

Remarks:

1. The DAMPING entry is referenced by the RSDAMP or SEDAMP Case Control commands. If a DAMPING entry is selected in the Case Control, the DAMPING values, including defaults, will override parameter inputs.
2. All damping selections are cumulative.
3. Structural damping specified by the G and GEFACT entries will replace any structural damping by:

$$\text{new[K4]} = \text{G[K]} + \text{GEFACT[K4]}$$

The GFACT value used is  $\text{GEFACT} = \text{GE} (\text{default}=0.0) * \text{GEFACT} (\text{default}=1.0)$  where GE is the structural damping value on the material entry.

4. Rayleigh viscous damping is calculated as:

$$[\text{B}]_{\text{Rayleigh}} = \alpha_1[\text{M}] + \alpha_2[\text{K}]$$

5. Hybrid damping on the residual structure is only active for direct solution sequences. For modal solution sequences, the SDAMP Case Control request should be used.
6. Hybrid damping for superelements uses modes that are calculated using the superelement mass and stiffness matrices before upstream superelements are added and before SPC or MPC constraints are imposed. These matrices are known as the JJ-type matrices.
7. The W3 and W4 values are used in transient response only. A zero value (default) will result in no damping. The equivalent viscous damping is calculated as:

$$[\text{B}]_{\text{equiv}} = \left(\frac{\text{G}}{\text{W3}}\right)[\text{K}] + \left(\frac{\text{GEFACT}}{\text{W4}}\right)[\text{K4}] + \left(\frac{1}{\text{WH}}\right)[\text{KH}]$$

8. For rotordynamic analyses (RGYRO in the Case Control Section), the DAMPING calculations for the residual structure (RSDAMP) are performed without any rotor contributions (support structure only).
9. Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

### Examples:

1. Specify a structural damping coefficient of 0.03 for the residual structure for frequency response.

Case Control:

RSDAMP=100

Bulk Data:

DAMPING, 100, 0.03

2. Specify a structural damping coefficient of 0.03 for the residual structure for transient response (use a “natural” frequency of 100 Hz)

Case Control:

RSDAMP=100

Bulk Data:

```
DAMPING, 100, 0.03,  
, 628.3
```

3. Specify hybrid damping for superelement 1. Use modal damping of two percent critical for the first 6 modes.

Case Control:

```
SUBCASE 1  
SUPER=1  
SEDAMP=100
```

Bulk Data:

```
DAMPING, 100, , , , 101  
HYBDAMP, 101, 102, 1001  
EIGRL, 102, , , 6  
TABDMP1, 1001, CRIT,  
, 0.0, 0.02, 1000.0, 0.02, ENDT
```

**DAREA****Load Scale Factor**

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with ACSRCE, RLOADi and TLOADi entries.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-------|-----|----|----|----|----|----|----|---|----|
| DAREA | SID | P1 | C1 | A1 | P2 | C2 | A2 |   |    |

**Example:**

|       |   |   |   |     |    |   |      |  |  |
|-------|---|---|---|-----|----|---|------|--|--|
| DAREA | 3 | 6 | 2 | 8.2 | 15 | 1 | 10.1 |  |  |
|-------|---|---|---|-----|----|---|------|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Identification number. (Integer > 0)  |
| Pi         | Grid, extra, or scalar point identification number. (Integer > 0)                             |
| Ci         | Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point.) |
| Ai         | Scale (area) factor. (Real)   |

**Remarks:**

1. One or two scale factors may be defined on a single entry.
2. Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.
3. Component numbers refer to the displacement coordinate system.
4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.
5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.
6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE/MOMENT/SLOAD entries (as appropriate) *if there are no LSEQ Bulk Data entries*.
7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements *if there are no LSEQ Bulk Data entries*.
8. In static analysis, DAREA entries may be used *only if there are no LSEQ Bulk Data entries*. They are ignored if there are any LSEQ Bulk Data entries.

**DCONADD****Design Constraint Set Combination**

Defines the design constraints for a subcase as a union of DCONSTR entries.

**Format:**

| 1       | 2    | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|------|--------|-----|-----|-----|-----|-----|-----|----|
| DCONADD | DCID | DC1    | DC2 | DC3 | DC4 | DC5 | DC6 | DC7 |    |
|         | DC8  | -etc.- |     |     |     |     |     |     |    |

**Example:**

|         |    |   |    |  |  |  |  |  |  |
|---------|----|---|----|--|--|--|--|--|--|
| DCONADD | 10 | 4 | 12 |  |  |  |  |  |  |
|---------|----|---|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| DCID       | Design constraint set identification number. (Integer > 0) |
| DCi        | DCONSTR entry identification number. (Integer > 0)         |

**Remarks:**

1. The DCONADD entry is selected by a DESSUB or DESGLB Case Control command.
2. All DCi must be unique from other DCi.
3. For PART SE, DCi from different PART SEs can be referenced on a single DCONADD and only DCONADD in the main Bulk Data Section, starts with 'BEGIN BULK' or 'BEGIN SUPER=0', will be considered as part of design constraints. Note that DCONADD entries in 'BEGIN SUPER=seid' where seid>0 will be ignored.

**DCONSTR**

## Design Constraints

Defines design constraints.

**Format:**

| 1       | 2    | 3   | 4          | 5          | 6     | 7      | 8 | 9 | 10 |
|---------|------|-----|------------|------------|-------|--------|---|---|----|
| DCONSTR | DCID | RID | LALLOW/LID | UALLOW/UID | LOWFQ | HIGHFQ |   |   |    |

**Example:**

|         |    |   |      |  |  |  |  |  |  |
|---------|----|---|------|--|--|--|--|--|--|
| DCONSTR | 10 | 4 | 1.25 |  |  |  |  |  |  |
|---------|----|---|------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| DCID       | Design constraint set identification number. (Integer > 0)   |
| RID        | DRESPi entry identification number. (Integer > 0)  |
| LALLOW/LID | Lower bound on the response quantity or the set identification ID of a number of TABLEDi entry that supplies the lower bound as a function of frequency. (Real; Default = -1.0E20) |
| UALLOW/UID | Upper bound on the response quantity or the set identification ID of a number of TABLEDi entry that supplies the upper bound as a function of frequency. (Real; Default = 1.0E20)  |
| LOWFQ      | Low end of frequency range in Hertz (Real $\geq$ 0.0; Default = 0.0). See Remark 8.  |
| HIGHFQ     | High end of frequency range in Hertz (Real $\geq$ LOWFQ; Default = 1.0E+20). See Remark 8.   |

**Remarks:**

1. The DCONSTR entry may be selected in the Case Control Section by the DESSUB or DESGLB command.
2. DCID may be referenced by the DCONADD Bulk Data entry.
3. For a given DCID, the associated RID can be referenced only once.
4. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESPi entry. If RID refers to an “EIGN” response, then the imposed bounds must be expressed in units of eigenvalue, (radian/time)<sup>2</sup>. If RID refers to a “FREQ” response, then the imposed bounds must be expressed in cycles/time.
5. LALLOW and UALLOW are unrelated to the stress limits specified on the MATi entry.
6. Constraints are computed as follows:

$$g = \frac{LALLOW - r}{GNORM} \text{ for lower bound constraints}$$

$$g = \frac{r - UALLOW}{GNORM} \text{ for upper bound constraints}$$

$$GNORM = \begin{cases} |LALLOW| \text{ for lower bounds if } |LALLOW| > GSCAL \\ |UALLOW| \text{ for upper bounds if } |UALLOW| > GSCAL \\ GSCAL \text{ otherwise} \end{cases}$$

GSCAL is specified on the DOPTPRM entry (Default = 0.001)

7. As Remark 6. indicates, small values of UALLOW and LALLOW require special processing and should be avoided. Bounds of exactly zero are particularly troublesome. This can be avoided by using a DRESP2 entry that offsets the constrained response from zero.
8. LOWFQ and HIGHFQ fields are functional only for RTYPE with a 'FR' or a 'PSD' prefix, e.g., FRDISP or on DRESP2 or DRESP3 entries that inherit the frequency value from these RTYPES. The bounds provided in LALLOW and UALLOW are applicable to a response only when the value of forcing frequency of the response falls between the LOWFQ and HIGHFQ. If the ATTB field of the DRESP1 entry is not blank and these responses are subsequently referenced on a DRESP2 or DRESP3, the LOWFQ and HIGHFQ are ignored when the DCONSTR refers to these DRESP2/DRESP3 responses. LOWFQ/HIGHFQ are not ignored when the DCONSTR refers to a DRESP1 that has a real number in the ATTB field.
9. LID and UID are optional inputs that identify tabular input to specify the lower and upper bounds as a function of frequency. They are applicable to the 'FR' and 'PSD' responses of Remark 8. and to DRESP2 and DRESP3 responses that inherit the frequency value from these RTYPES.
10. For PART SE, DCONSTR entries can reside in each individual PART SE Bulk Data Section starting with 'BEGIN SUPER=seid'. If DCID is different from a PART SE to the next, DCONADD in the residual (or main Bulk Data Section) can be utilized to group DCONSTR entries together for a single subcase.

**DDVAL****Discrete Design Variable Values**

Defines real, discrete design variable values for use in discrete variable optimization, topometry optimization or rotordynamics.

Format:

| 1     | 2  | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|-------|----|-------|-------|-------|-------|-------|-------|-------|----|
| DDVAL | ID | DVAL1 | DVAL2 | DVAL3 | DVAL4 | DVAL5 | DVAL6 | DVAL7 |    |

Alternate Format:

|       |    |       |      |       |    |     |  |  |  |
|-------|----|-------|------|-------|----|-----|--|--|--|
| DDVAL | ID | DVAL1 | THRU | DVAL2 | BY | INC |  |  |  |
|-------|----|-------|------|-------|----|-----|--|--|--|

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

|  |       |       |        |        |        |  |  |  |  |
|--|-------|-------|--------|--------|--------|--|--|--|--|
|  | DVAL8 | DVAL9 | DVAL10 | DVAL11 | -etc.- |  |  |  |  |
|--|-------|-------|--------|--------|--------|--|--|--|--|

Continuation Entry Format 2:

|  |       |      |       |    |     |  |  |  |  |
|--|-------|------|-------|----|-----|--|--|--|--|
|  | DVAL8 | THRU | DVAL9 | BY | INC |  |  |  |  |
|--|-------|------|-------|----|-----|--|--|--|--|

Example:

| 1     | 2   | 3    | 4   | 5   | 6    | 7   | 8   | 9 | 10 |
|-------|-----|------|-----|-----|------|-----|-----|---|----|
| DDVAL | 110 | 0.1  | 0.2 | 0.3 | 0.5  | 0.6 | 0.4 |   |    |
|       | .7  | THRU | 1.0 | BY  | 0.05 |     |     |   |    |
|       | 1.5 | 2.0  |     |     |      |     |     |   |    |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique set identification number (Integer > 0) See Remark 1.    |
| DVALi      | Discrete values (Real, or THRU or BY) See Remarks 2. through 5. |
| INC        | Discrete value increment (Real) See Remarks 2., 4., 5.          |

Remarks:

1. DDAVAL entries may be referenced by a DESVAR entry in its DDVAL field (field 8), a TOMVAR entry in its DSVID field, a entry in its DDVALID field (field 4) or a RSPINR entry in its SPTID field (field 6).

2. Trailing fields on a DDVAL record can be left blank if the next record is of type DVALi THRU DVALj BY INC. Also fields 7 - 9 must be blank when the type DVALi THRU DVALj BY INC is used in fields 2 - 6 and fields 8 - 9 must be blank when the type DVALi "THRU" DVALj "BY" INC is used in fields 3 - 7 for the first record. Embedded blanks are not permitted in other cases.
3. The DVALi values in a sequence need not be in an ascending or descending order. They can be in any order.
4. If the format DVALi THRU DVALj BY INC is employed, INC must be a positive real value if DVALi < DVALj and INC must be a negative real value if DVALi > DVALj. The program terminates the execution with an appropriate fatal error if this requirement is not satisfied.
5. The format DVALi THRU DVALj BY INC defines a list of discrete values given by DVALi, DVALi+INC, DVALi+2.0\*INC, ..., DVALj. The last discrete value of DVALj is always included, even if the range is not evenly divisible by INC.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**DEACTEL**

## Elements To Be Deactivated for SOL 400 Analysis

This entry identifies the elements that do not participate in the analysis or do not participate in a particular physics pass of a coupled analysis. SOL 400 for NLSTATIC, and NLTRAN, as well Perturbation analyses only.

**Format:**

| 1       | 2  | 3 | 4 | 5 | 6    | 7 | 8 | 9 | 10 |
|---------|----|---|---|---|------|---|---|---|----|
| DEACTEL | ID |   |   |   | ISET |   |   |   |    |

**Example:**

|         |   |  |  |  |     |  |  |  |  |
|---------|---|--|--|--|-----|--|--|--|--|
| DEACTEL | 2 |  |  |  | 200 |  |  |  |  |
|---------|---|--|--|--|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | ID of the DEACTEL entry. This is referenced by a DEACTEL=ID Case Control command. (Integer; no Default) |
| ISET       | SID of a list of elements defined by a SET3 entry. (Integer; no Default)                                |

**Remarks:**

1. DEACTEL can only be referenced before the first subcase and/or inside the first subcase and/or inside the first step of the first subcase and/or inside each substep of the first step of the first subcase. References that are made anywhere else are currently ignored. It is referenced by the Case Control command DEACTEL=ID.
2. The description field (DES) of the SET3 entry referenced by ISET can only be of type ELEM.
3. Element IDs in the list originating from some set that do not exist are ignored.

**DEACTEL**

## Define Elements that Should be Deactivated for a Particular Subcase in SOL 600

This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase. Some or all of these elements can be re-activated in a subsequent subcase using the ACTIVAT entry.

**Format:**

| 1       | 2  | 3      | 4      | 5     | 6    | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|-------|------|---|---|---|----|
| DEACTEL | ID | STRESS | STRAIN | IGEOM | ISET |   |   |   |    |

**Example:**

|         |   |   |   |   |     |  |  |  |  |
|---------|---|---|---|---|-----|--|--|--|--|
| DEACTEL | 2 | 1 | 1 | 0 | 200 |  |  |  |  |
|---------|---|---|---|---|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number of a matching DEACTEL Case Control command defining the subcase to which these elements should be deactivated. (Integer; no Default)  |
| STRESS     | Flag to control output stresses after the elements have been deactivated. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0      Stresses have the same value as just prior to deactivation</li> <li>1      Stresses are set to zero</li> </ul> |
| STRAIN     | Flag to control output strains after the elements have been deactivated. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0      Strains have the same value as just prior to deactivation</li> <li>1      Strains are set to zero</li> </ul>    |
| IGEOM      | Flag to control whether to update the geometry on the post file. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0      Update the geometry.</li> <li>1      Do not update the geometry.</li> </ul>   |
| ISET       | ID of a list of elements described by SET3 is ID=ISET to be deactivated. (Integer; no Default)  |

**Remarks:**

1. This entry maps to Marc's DEACTIVATE (option A) History definition (option B is not supported in SOL 600).

**DEFORM****Static Element Deformation**

Defines enforced axial deformation for one-dimensional elements for use in statics problems.

**Format:**

| 1      | 2   | 3    | 4  | 5    | 6  | 7    | 8  | 9 | 10 |
|--------|-----|------|----|------|----|------|----|---|----|
| DEFORM | SID | EID1 | D1 | EID2 | D2 | EID3 | D3 |   |    |

**Example:**

|        |   |     |     |     |      |  |  |  |  |
|--------|---|-----|-----|-----|------|--|--|--|--|
| DEFORM | 1 | 535 | .05 | 536 | -.10 |  |  |  |  |
|--------|---|-----|-----|-----|------|--|--|--|--|

| Descriptor       | Meaning  |
|------------------|--|
| SID              | Deformation set identification number. (Integer > 0)       |
| EID <sub>i</sub> | Element number. (Integer > 0)                              |
| D <sub>i</sub>   | Deformation. (Real; positive value represents elongation.) |

**Remarks:**

1. The referenced element must be one-dimensional (CROD, CONROD, CTUBE, CBAR, CBEAM).
2. Deformation sets must be selected in the Case Control Section with DEFORM = SID.
3. One to three enforced element deformations may be defined on a single entry.
4. The DEFORM entry, when called by the DEFORM Case Control command, is applicable to linear static, inertia relief, differential stiffness, and buckling (Solutions 101, 105, 114, and 200) and will produce fatal messages in other solution sequences. Use SPCD to apply enforced displacements in solution sequences for which DEFORM does not apply.

**DEFUSET**

## Degree-of-Freedom Set Name Definition

Defines new names for degree-of-freedom sets.

**Format:**

| 1       | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|---------|------|------|------|------|------|------|------|------|----|
| DEFUSET | OLD1 | NEW1 | OLD2 | NEW2 | OLD3 | NEW3 | OLD4 | NEW4 |    |

**Example:**

|         |    |   |    |   |    |   |  |  |  |
|---------|----|---|----|---|----|---|--|--|--|
| DEFUSET | U2 | X | U4 | Y | U3 | Z |  |  |  |
|---------|----|---|----|---|----|---|--|--|--|

| Descriptor | Meaning                                    |
|------------|--|
| OLDi       | Default set name. (One to four characters) |
| NEWi       | New set name. (One to four characters)     |

**Remarks:**

1. From one to four set names may be specified on a single entry.
2. OLDi must refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#). It is recommended that OLDi refer only to the set names U1 through U6. If sets PA or PS are referenced, a user fatal message is issued.
3. All NEWi names must be unique with respect to all other set names.
4. The DEFUSET entry is optional since default set names exist for all displacement sets.
5. The DEFUSET entry must be present in the Bulk Data Section in all restarts.

**DELAY****Dynamic Load Time Delay**

Defines the time delay term  $\tau$  in the equations of the dynamic loading function.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-------|-----|----|----|----|----|----|----|---|----|
| DELAY | SID | P1 | C1 | T1 | P2 | C2 | T2 |   |    |

**Example:**

|       |   |    |   |      |   |   |     |  |  |
|-------|---|----|---|------|---|---|-----|--|--|
| DELAY | 5 | 21 | 6 | 4.25 | 7 | 6 | 8.1 |  |  |
|-------|---|----|---|------|---|---|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Identification number of the DELAY entry. (Integer > 0)   |
| Pi         | Grid, extra, or scalar point identification number. (Integer > 0)                                   |
| Ci         | Component number. (Integer 1 through 6 for grid point, blank or 0 for extra point or scalar point.) |
| Ti         | Time delay $\tau$ for designated point Pi and component Ci. (Real)                                  |

**Remarks:**

1. One or two dynamic load time delays may be defined on a single entry.
2. SID must also be specified on a RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entry. See those entry descriptions for the formulas that define the manner in which the time delay  $\tau$  is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DELAY entries may only be applied to loads on points in the residual structure.

## DEQATN

## Equation Definition

Defines one or more equations for use in analysis.

Format:

| 1      | 2    | 3 | 4 | 5 | 6                | 7 | 8 | 9 | 10 |
|--------|------|---|---|---|------------------|---|---|---|----|
| DEQATN | EQID |   |   |   | EQUATION         |   |   |   |    |
|        |      |   |   |   | EQUATION (Cont.) |   |   |   |    |

Example:

|        |    |  |  |
|--------|----|--|--|
| DEQATN | 14 | F1(A, B, C, D, R) = A + B · C - (D ** 3 + 10.0) + sin(PI(1) · R) |  |
|        |    | + A**2/(B - C) ; F = A + B - F1 · D                              |  |

| Descriptor | Meaning  |
|------------|--|
| EQID       | Unique equation identification number. (Integer > 0) |
| EQUATION   | Equation(s). See Remarks. (Character)                |

Remarks:

1. EQUATION is a single equation or a set of nested equations and is specified in fields 3 through 9 on the first entry and may be continued on fields 2 through 9 on the continuation entries. On the continuation entries, no commas can appear in columns 1 through 8. All data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

A single equation has the following format:

variable-1 (x1, x2, ..., xn)=expression-1

A set of nested equations is separated by semicolons and has the format:

variable-1 (x1, x2, ..., xn)=expression-1;  
variable-2=expression-2;variable-3=expression-3;

etc.

variable-m=expression-m

Expression-i is a collection of constants, real variables, and real functions, separated by operators, and must produce a single real value. (x1, x2, ..., xn) is the list of all the variable names (except variable-i) that appear in all expressions. Variable-i may be used in subsequent expressions. The last equation, variable-m=expression-m, provides the value that is returned to the Bulk Data entry that references EQID; e.g., DRESP2. The example above represents the following mathematical equations:

$$F1 = A + B \cdot C - (D^3 + 10) + \sin(\pi \cdot R) + \frac{A^2}{B - C}$$

$$F = A + B + F1 \cdot D$$

where SIN and PI are intrinsic functions. See Remark 4.

2. EQUATION may contain embedded blanks. EQUATION must contain less than 32,000 nonblank characters. If more characters are required for use with a DRESP2 entry, the DRESP2 can be divided into two or more DRESP2 entries with a master DRESP2 referencing subsequent DRESP2s.
3. The syntax of the expressions follows FORTRAN language standards. The allowable arithmetic operations are shown in [Table 8](#) in the order of execution precedence. Parenthesis are used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses.

**Table 8** DEQATN Entry Operators

| Operator | Operation   | Sample Expressions | Interpreted As |
|----------|---|--------------------|----------------|
| -, +     | Negative or Positive immediately preceded by exponentiation | X ** -Y            | X ** (-Y)      |
| **       | Exponentiation  | -X ** Y            | (-X) ** Y      |
| -, +     | Negative or Positive  | -X-Y               | (-X)-Y         |
| *, /     | Multiplication or Division                                  | X * Y-Z            | (X * Y)-Z      |
| +, -     | Addition or Subtraction                                     | X+Y                | X+Y            |

4. The expressions may contain intrinsic functions. [Table 9](#) contains the format and descriptions of functions that may appear in the expressions. The use of functions that may be discontinuous must be used with caution because they can cause discontinuous derivatives. These are ABS, DIM, MAX, MIN, and MOD. For examples and further details see the [MSC Nastran DMAP Programmer's Guide](#).

**Table 9** DEQATN Entry Functions

| Format      | Description                       | Mathematical Expressions |
|-------------|-----------------------------------|--------------------------|
| ABS(x)      | absolute value                    | $ x $                    |
| ACOS(x)     | arccosine                         | $\cos^{-1} x$            |
| ACOSH(x)    | hyperbolic arccosine              | $\cosh^{-1} x$           |
| ASIN(x)     | arcsine                           | $\sin^{-1} x$            |
| ASINH(x)    | hyperbolic arcsine                | $\sinh^{-1} x$           |
| ATAN(x)     | arctangent                        | $\tan^{-1} x$            |
| ATAN2(x,y)  | arctangent of quotient            | $\tan^{-1} (x/y)$        |
| ATANH(x)    | hyperbolic arctangent             | $\tanh^{-1} x$           |
| ATANH2(x,y) | hyperbolic arctangent of quotient | $\tanh^{-1} (x/y)$       |

Table 9 DEQATN Entry Functions (continued)

| Format  | Description                           | Mathematical Expressions  |
|---|---------------------------------------|---|
| AVG(X <sub>1</sub> , X <sub>2</sub> , .., X <sub>n</sub> )  | average                               | $\frac{1}{n} \sum_{i=1}^n X_i$  |
| COS(x)  | cosine                                | cos x   |
| COSH(x)   | hyperbolic cosine                     | cosh x  |
| DB(P, PREF)   | sound pressure in decibel             | $20.0 \cdot \log\left(\frac{P}{\text{PREF}}\right)$   |
| DBA(P, PREF, F)   | sound pressure in decibel (perceived) | $20.0 \cdot \log\left(\frac{P}{\text{PREF}}\right) + 10.0 \cdot \log(\text{Ta1}) + 10.0 \cdot \log(\text{Ta2})$ |
| DIM(x,y)  | positive difference                   | x-MIN(x,y)  |
| EXP(x)  | exponential                           | e <sup>x</sup>  |
| INVDB(DB, PREF)   | inverse Db                            | $10^{\left(\frac{DB}{20.0} + \log PREF\right)}$   |
| INVDAB(DBA, PREF, F)  | inverse Dba                           | $10^{\left(\frac{DBA - 10.0 \cdot \log(\text{Ta1}) - 10.0 \cdot \log(\text{Ta2})}{20.0}\right)}$                |
| LOG(x)  | natural logarithm                     | log <sub>e</sub> x  |
| LOG10(x)  | common logarithm                      | log <sub>10</sub> x   |
| LOGX(x,y)   | base x logarithm                      | log <sub>x</sub> y  |
| MAX(x <sub>1</sub> , x <sub>2</sub> , ...)                  | maximum                               | maximum of x <sub>1</sub> , etc.  |
| MIN(x <sub>1</sub> , x <sub>2</sub> , ...)                  | minimum                               | minimum of x <sub>1</sub> , etc.  |
| MOD(x,y)  | remainder (modulo)                    | x - y · (INT(x/y))  |
| PI(x)   | multiples of pi ( $\pi$ )             | x · $\pi$   |
| RSS(X <sub>1</sub> , X <sub>2</sub> , ..., X <sub>n</sub> ) | square root of sum of squares         | $\sqrt{\sum_{i=1}^n X_i^2}$   |
| SIN(x)  | sine                                  | sin x   |
| SINH(x)   | hyperbolic sine                       | sinh x  |
| SQRT(x)   | square root                           | $\sqrt{x}$  |

Table 9 DEQATN Entry Functions (continued)

| Format  | Description        | Mathematical Expressions |
|---|--------------------|--------------------------|
| SSQ(X <sub>1</sub> , X <sub>2</sub> , ..., X <sub>n</sub> ) | sum of squares     | $\sum_{i=1}^n X_i^2$     |
| SUM(X <sub>1</sub> , X <sub>2</sub> , ..., X <sub>n</sub> ) | summation          | $\sum_{i=1}^n X_i$       |
| TAN(x)  | tangent            | $\tan x$                 |
| TANH(x)   | hyperbolic tangent | $\tanh x$                |

where

|  |   |  |
|--|---|--|
| X <sub>1</sub> , X <sub>2</sub> , .., X <sub>n</sub> , P | = | structure responses or acoustic pressure               |
| PREF   | = | reference pressure                                     |
| F  | = | forcing frequency                                      |
| DB   | = | acoustic pressure in Decibel                           |
| DBA  | = | perceived acoustic pressure in Decibel                 |
| Ta1  | = | $\frac{K_3 \cdot F^4}{(F^2 + P_2^2)(F^2 + P_3^2)}$     |
| Ta2  | = | $\frac{K_1 \cdot F^4}{(F^2 + P_1^2)^2(F^2 + P_4^2)^2}$ |
| K1   | = | 2.242882e+16   |
| K3   | = | 1.562339   |
| P1   | = | 20.598997  |
| P2   | = | 107.65265  |
| P3   | = | 737.86223  |
| P4   | = | 12194.22   |

5. If the DEQATN entry is referenced by the:

- DVCREL2, DVMREL2, or DVPREL2 entry, then X<sub>i</sub> represents the DVIDj and LABLk fields.
- DRESP2 entry, then X<sub>i</sub> represents the DVIDj, LABLk, NRm, Gp, DPIPq, DCICr, DMIMs, DPI2Pt, DCI2Cu, DMI2Mv, and NRRw fields in that order.
- GMLOAD, GMBC, or TEMPF entries, then
  - X<sub>1</sub> represents x in the basic coordinate system,
  - X<sub>2</sub> represents y in the basic coordinate system, and
  - X<sub>3</sub> represents z in the basic coordinate system.

- GMCURV entry, then  
 $X_1$  represents line parameter u.
  - GMSURF entry, then  
 $X_1$  represents surface parameter u and  
 $X_2$  represents surface parameter v.
6. If the DEQATN entry is referenced by the GMLOAD, GMBC, TEMPF, GMCURV, or GMSURF entries and your computer has a short word length (e.g., 32 bits/word), then EQUATION is processed with double precision and constants may be specified in double precision; e.g., 1.2D0. If your machine has a long word length (e.g., 64 bits/word) then EQUATION is processed in single precision and constants must be specified in single precision; e.g., 1.2. If the DEQATN entry is referenced by DRESP2, DVCREL2, DVMREL2 or DVPREL2 entries, constants must be specified in single precision regardless of your machine's word length.
  7. The DMAP logical operators NOT, AND, OR, XOR, and XQV cannot be used as  $X_i$  names.
  8. Input errors on the DEQATN entry often result in poor messages. Substituting a "[" for a parenthesis or violating the restriction against large field format are examples. Known messages are UFM 215, SFM 233 and UFM 5199. If any of these messages are encountered then review the DEQATN entry input.
  9. Intrinsic functions AVG, MAX, MIN, RSS, SSQ and SUM are limited to <97 arguments. If more arguments are desired, the functions may be divided up. For example, to perform a sum of squares on 170 items, use:  
$$\text{SSQ}(X1,X2,\dots,X95) + \text{SSQ}(X96,X97,\dots,X170).$$
For MAX/MIN, the following concatenation can be used:  
$$\text{MAX1} = \text{MAX}(X1,X2,\dots,X95), \text{MAX2} = \text{MAX}(X96,X97\dots X170); \text{MAXT} = \text{MAX}(\text{MAX1}, \text{MAX2}).$$
For AVG, an example to average 170 terms is:  
$$\text{AVG1} = \text{AVG}(X1,X2,\dots,X95); \text{AVG2} = \text{AVG}(X96,X97\dots X170); \text{AVGT} = (95. * \text{AVG1} + 75. * \text{AVG2}) / 170.$$
  10. Arithmetic is carried out using the type of the input data. For example, in the expression:

$$X = A^{**}(1/2)$$

both values in the exponent are integers so that the value returned for the exponent is calculated using integer arithmetic or  $1/2 = 0$ . In this case  $1/2$  should be replaced by  $(.5)$ .

**DESVAR****Design Variable**

Defines a design variable for design optimization.

**Format:**

| 1      | 2  | 3     | 4     | 5   | 6   | 7     | 8     | 9 | 10 |
|--------|----|-------|-------|-----|-----|-------|-------|---|----|
| DESVAR | ID | LABEL | XINIT | XLB | XUB | DELXV | DDVAL |   |    |

**Example:**

|        |   |       |      |     |      |     |  |  |  |
|--------|---|-------|------|-----|------|-----|--|--|--|
| DESVAR | 2 | BARA1 | 35.0 | 10. | 100. | 0.2 |  |  |  |
|--------|---|-------|------|-----|------|-----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique design variable identification number. (Integer > 0)  |
| LABEL      | User-supplied name for printing purposes. (Character)  |
| XINIT      | Initial value. (Real; $\text{XLB} \leq \text{XINIT} \leq \text{XUB}$ )   |
| XLB        | Lower bound. (Real; Default = $-1.0\text{E}+20$ )  |
| XUB        | Upper bound. (Real; Default = $+1.0\text{E}+20$ )  |
| DELXV      | Fractional change allowed for the design variable during approximate optimization. (Real > 0.0; for Default see Remark 2.)                                 |
| DDVAL      | ID of a DDVAL entry that provides a set of allowable discrete values. (Blank or Integer > 0; Default=blank for continuous design variables. See Remark 3.) |

**Remarks:**

1. DELXV can be used to control the change in the design variable during one optimization cycle.
2. If DELXV is blank, the default is taken from the specification of the DELX parameter on the DOPTPRM entry. If DELX is not specified, then the default is 0.5.
3. If the design variable is to be discrete (Integer > 0 in DDVAL field), and if either of the XLB and/or XUB bounds are wider than those given by the discrete list of values on the corresponding DDVAL entry, XLB and/or XUB will be replaced by the minimum and maximum discrete values.

**DETSPH****Spherical Detonation Wave**

Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials. Used in SOL 700 only.

Format:

| 1      | 2   | 3   | 4 | 5 | 6 | 7   | 8    | 9 | 10 |
|--------|-----|-----|---|---|---|-----|------|---|----|
| DETSPH | DID | MID | X | Y | Z | VEL | TIME |   |    |

Example:

|        |     |    |      |       |      |       |        |  |  |
|--------|-----|----|------|-------|------|-------|--------|--|--|
| DETSPH | 100 | 10 | 96.5 | 177.6 | 37.4 | 2379. | 1.7E-6 |  |  |
|--------|-----|----|------|-------|------|-------|--------|--|--|

| Descriptor | Meaning  |
|------------|--|
| DID        | Unique detonation number. (Integer > 0; Required)                        |
| MID        | References MATDEUL id of the exploding material. (Integer > 0; Required) |
| X, Y, Z    | Coordinates of the ignition point. (Real, 0.0)                           |
| VEL        | Velocity of the detonation wave. (Real $\geq 0.0$ , 0.0)                 |
| TIME       | Detonation time. (Real $\geq 0.0$ , 0.0)                                 |

Remark:

1. An element detonates when a spherical detonation wave originating from the detonation point at the specified time reaches the element.

**DIVERG****Divergence Analysis Data**

Defines Mach numbers (m) for a static aeroelastic divergence analysis.

**Format:**

| 1      | 2   | 3     | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-------|--------|----|----|----|----|----|----|
| DIVERG | SID | NROOT | M1     | M2 | M3 | M4 | M5 | M6 |    |
|        | M7  | M8    | -etc.- |    |    |    |    |    |    |

**Example:**

|        |    |   |    |    |    |  |  |  |  |
|--------|----|---|----|----|----|--|--|--|--|
| DIVERG | 70 | 2 | .5 | .8 | .9 |  |  |  |  |
|--------|----|---|----|----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique set identifier. (Integer > 0)  |
| NROOT      | Number of divergence roots that are to be output and their eigenvectors printed. (Integer; Default = 1) |
| Mi         | Mach number. (Real $\geq 0.0$ )   |

**Remarks:**

1. The DIVERG entry is referenced in Case Control by “DIVERG = SID”.
2. The NROOT lowest divergence dynamic pressures are printed. If there are fewer than NROOT pressures, all available dynamic pressures are printed.
3. Mi values must be distinct.
4. A blank Mach number field terminates the input.

**DLINK****Multiple Design Variable Linking**

Relates one design variable to one or more other design variables.

**Format:**

| 1     | 2    | 3     | 4      | 5     | 6    | 7  | 8    | 9  | 10 |
|-------|------|-------|--------|-------|------|----|------|----|----|
| DLINK | ID   | DDVID | C0     | CMULT | IDV1 | C1 | IDV2 | C2 |    |
|       | IDV3 | C3    | -etc.- |       |      |    |      |    |    |

**Example:**

|       |    |     |     |      |   |     |   |      |  |
|-------|----|-----|-----|------|---|-----|---|------|--|
| DLINK | 10 | 2   | 0.1 | 0.33 | 4 | 2.0 | 6 | -1.0 |  |
|       | 8  | 7.0 |     |      |   |     |   |      |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)                      |
| DDVID      | Dependent design variable identification number. (Integer > 0)   |
| C0         | Constant term. (Real; Default = 0.0)                             |
| CMULT      | Constant multiplier. (Real; Default = 1.0)                       |
| IDVi       | Independent design variable identification number. (Integer > 0) |
| Ci         | Coefficient i corresponding to IDVi. (Real)                      |

**Remarks:**

1. DLINK defines the relationship

$$\text{DDVID} = \text{C0} + \text{CMULT} * \sum_i \text{Ci} * \cdot \text{IDVi}$$

2. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
3. CMULT provides a simple means of scaling the Ci. For example if Ci = 1/7, 2/7, 4/7, etc. is desired, then CMULT = 1/7 and Ci = 1, 2, 4, etc., may be input.
4. An independent IDVi must not occur on the same DLINK entry more than once.
5. ID is for user reference only.
6. If a design variable is specified as dependent on a DLINK entry, then it cannot be specified as independent on another DLINK entry.

**DLOAD**

## Dynamic Load Combination or Superposition

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

Format:

| 1     | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|-----|----|--------|----|----|----|----|----|----|
| DLOAD | SID | S  | S1     | L1 | S2 | L2 | S3 | L3 |    |
|       | S4  | L4 | -etc.- | *  |    |    |    |    |    |

Examples:

|       |      |     |     |   |      |   |     |   |  |
|-------|------|-----|-----|---|------|---|-----|---|--|
| DLOAD | 17   | 1.0 | 2.0 | 6 | -2.0 | 7 | 2.0 | 8 |  |
|       | -2.0 | 9   |     |   |      |   |     |   |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. See Remarks 1. and 4. (Integer > 0)  |
| S          | Scale factor. See Remarks 2. and 8. (Real)   |
| Si         | Scale factors. See Remarks 2., 7. and 8. (Real)  |
| Li         | Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries. See Remarks 3. and 7. (Integer > 0) |

Remarks:

1. Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
  2. The load vector being defined by this entry is given by
- $$\{P\} = S \sum_i S_i \{P_i\}$$
3. Each Li must be unique from any other Li on the same entry.
  4. SID must be unique from all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries.
  5. Nonlinear transient load sets (NOLINI entries) may not be specified on DLOAD entries. NOLINI entries are selected separately in the Case Control Section by the NONLINEAR command.
  6. A DLOAD entry may not reference a set identification number defined by another DLOAD entry.
  7. The scale factor Si will apply to all dynamic load entries with load set identification number of Li.
  8. For RC network solver in thermal analysis, the S and Si factors are always taken as 1.
  9. If Modules are present then this entry may only be specified in the main Bulk Data section.

**DMI****Direct Matrix Input**

Defines matrix data blocks. Generates a matrix of the following form:

$$[\text{NAME}] = \begin{bmatrix} X_{11} & X_{12} & \dots & X_{1n} \\ X_{21} & X_{22} & \dots & X_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ X_{m1} & \dots & \dots & X_{mn} \end{bmatrix}$$

where the elements  $X_{ij}$  may be real ( $X_{ij} = A_{ij}$ ) or complex ( $X_{ij} = A_{ij} + iB_{ij}$ ). The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

| 1   | 2    | 3   | 4    | 5   | 6    | 7 | 8 | 9 | 10 |
|-----|------|-----|------|-----|------|---|---|---|----|
| DMI | NAME | "0" | FORM | TIN | TOUT |   | M | N |    |

**Column Entry Format for Real Matrices:**

|     |      |         |    |         |           |  |        |    |  |
|-----|------|---------|----|---------|-----------|--|--------|----|--|
| DMI | NAME | J       | I1 | A(I1,J) | A(I1+1,J) |  | -etc.- | I2 |  |
|     |      | A(I2,J) |    | -etc.-  |           |  |        |    |  |

**Column Entry Format for Complex Matrices:**

|     |      |    |         |         |         |           |           |        |  |
|-----|------|----|---------|---------|---------|-----------|-----------|--------|--|
| DMI | NAME | J  | I1      | A(I1,J) | B(I1,J) | A(I1+1,J) | B(I1+1,J) | -etc.- |  |
|     |      | I2 | A(I2,J) | B(I2,J) | -etc.-  |           |           |        |  |

**Example of a Real Matrix:**

|     |     |   |   |    |    |    |   |   |  |
|-----|-----|---|---|----|----|----|---|---|--|
| DMI | BBB | 0 | 2 | 1  | 1  |    | 4 | 2 |  |
| DMI | BBB | 1 | 1 | 1. | 3. | 5. |   |   |  |
| DMI | BBB | 2 | 2 | 6. | 4  | 8. |   |   |  |

$$\text{BBB} = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

**Example of a Complex Matrix:**

|     |     |   |   |     |     |     |     |   |  |
|-----|-----|---|---|-----|-----|-----|-----|---|--|
| DMI | QQQ | 0 | 2 | 3   | 3   |     | 4   | 2 |  |
| DMI | QQQ | 1 | 1 | 1.0 | 2.0 | 3.0 | 0.0 | 3 |  |

|     |     |     |   |     |     |   |     |     |  |
|-----|-----|-----|---|-----|-----|---|-----|-----|--|
|     | 5.0 | 6.0 |   |     |     |   |     |     |  |
| DMI | QQQ | 2   | 2 | 6.0 | 7.0 | 4 | 8.0 | 9.0 |  |

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & 0.0 + 0.0i \\ 3.0 + 0.0i & 6.0 + 7.0i \\ 5.0 + 6.0i & 0.0 + 0.0i \\ 0.0 + 0.0i & 8.0 + 9.0i \end{bmatrix}$$

| Descriptor | Meaning  |
|------------|--|
| NAME       | Name of the matrix. See Remark 1. Name is used to reference the data block in the DMAP sequence. (One to eight alphanumeric characters, the first of which must be alphabetic.)  |
| FORM       | Form of matrix, as follows: (Integer)  |
|            | 1      Square matrix (not symmetric)<br>2      General rectangular matrix<br>3      Diagonal matrix (M=number of rows, N = 1)<br>4      Lower triangular factor<br>5      Upper triangular factor<br>6      Symmetric matrix<br>8      Identity matrix (M=number of rows, N = M) |
| TIN        | Type of matrix being input, as follows: (Integer)  |
|            | 1      Real, single precision (one field used/element)<br>2      Real, double precision (one field used/element)<br>3      Complex, single precision (two fields used/element)<br>4      Complex, double precision (two fields used/element)                                     |

| Descriptor   | Meaning  |
|--------------|--|
| TOUT         | Type of matrix being output, as follows: (Integer) <ul style="list-style-type: none"> <li>0 Set by precision cell</li> <li>1 Real, single precision</li> <li>2 Real, double precision</li> <li>3 Complex, single precision</li> <li>4 Complex, double precision</li> </ul> |
| M            | Number of rows in NAME. (Integer > 0)  |
| N            | Number of columns in NAME. Except for FORM 3 and 8. (Integer > 0)  |
| "0"          | Indicates the header entry.  |
| J            | Column number of NAME. (Integer > 0)   |
| I1, I2, etc. | Row number of NAME, which indicates the beginning of a group of nonzero elements in the column. See Remark 13. (Integer > 0)   |
| A(Ix,J)      | Real part of element (see TIN). (Real)   |
| B(Ix,J)      | Imaginary part of element (see TIN). (Real)  |

**Remarks:**

1. In order to use the DMI feature, the user must write a DMAP, or make alterations to a solution sequence that includes the DMIIN module. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. The total number of DMIs and DTIs may not exceed 1000.
3. Field 3 of the header entry must contain an integer of zero (0).
4. For symmetric matrices, the entire matrix must be input.
5. Only nonzero terms need be entered.
6. Leading and trailing zeros in a column do not have to be entered. However, a blank field between nonzero fields on this entry is not equivalent to a zero. If a zero input is required, the appropriate type zero must be entered (i.e., 0.0 or 0.0D0).
7. Complex input must have both the real and imaginary parts entered if either part is nonzero; i.e., the zero component must be input explicitly.
8. If A(Ix,J) is followed by "THRU" in the next field and an integer row number "IX" after the THRU, then A(Ix,J) will be repeated in each row through IX. The "THRU" must follow an element value. For example, the entries for a real matrix RRR would appear as follows:

| 1   | 2    | 3 | 4  | 5       | 6    | 7  | 8  | 9       | 10 |
|-----|------|---|----|---------|------|----|----|---------|----|
| DMI | NAME | J | I1 | A(I1,J) |      |    | I1 | A(I2,J) |    |
| DMI | RRR  | 1 | 2  | 1.0     | THRU | 10 | 12 | 2.0     |    |

These entries will cause the first column of the matrix RRR to have a zero in row 1, the values 1.0 in rows 2 through 10, a zero in row 11, and 2.0 in row 12.

9. Each column must be a single logical entry. The terms in each column must be specified in increasing row number order.
10. The “FORM” options 4, 5, and 8 are nonstandard forms and may be used only in conjunction with the modules indicated in [Table 10](#).

**Table 10 DMI FORM Options**

| FORM | Matrix Description      | Modules |     |        |       |
|------|-------------------------|---------|-----|--------|-------|
|      |                         | ADD     | FBS | MATPRN | MPYAD |
| 4    | Lower Triangular Factor |         | X   | X      |       |
| 5    | Upper Triangular Factor |         | X   | X      |       |
| 8    | Identity                | X       | X   | X      | X     |

11. Form 3 matrices are converted to Form 6 matrices, which may be used by any module.
12. Form 7 matrices may not be defined on this entry.
13. I1 must be specified. I2, etc. are not required if their matrix elements follow the preceding element in the next row of the matrix. For example, in the column entry for column 1 of QQQ, neither I2 nor I3 is specified.
14. The DMIG entry is more convenient for matrices with rows and columns that are referenced by grid or scalar point degrees-of-freedom.
15. If Modules are present then this entry may only be specified in the main Bulk Data section.

**DMDIA****Direct Matrix Input for Axisymmetric Analysis**

Defines axisymmetric (fluid or structure) related direct input matrix terms.

The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

| 1     | 2    | 3   | 4   | 5   | 6    | 7 | 8 | 9 | 10 |
|-------|------|-----|-----|-----|------|---|---|---|----|
| DMDIA | NAME | "0" | IFO | TIN | TOUT |   |   |   |    |

**Column Entry Format:**

|       |      |    |    |        |    |  |  |  |  |
|-------|------|----|----|--------|----|--|--|--|--|
| DMDIA | NAME | GJ | CJ | NJ     |    |  |  |  |  |
|       | G1   | C1 | N1 | A1     | B1 |  |  |  |  |
|       | G2   | C2 |    | -etc.- |    |  |  |  |  |

**Example:**

|       |      |    |   |        |        |  |  |  |  |
|-------|------|----|---|--------|--------|--|--|--|--|
| DMDIA | B2PP | 0  | 1 | 3      | 4      |  |  |  |  |
| DMDIA | B2PP | 32 |   |        |        |  |  |  |  |
|       | 1027 | 3  |   | 4.25+6 | 2.27+3 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Name of the matrix. See Remark 2. (One to eight alphanumeric characters, the first of which is alphabetic.) |
| IFO        | Form of matrix: (Integer)   |
|            | 1      Square matrix  |
|            | 2      General rectangular matrix   |
|            | 6      Symmetric matrix   |
| TIN        | Type of matrix being input: (Integer)   |
|            | 1      Real, single precision (One field is used per element.)  |
|            | 3      Complex, single precision (Two fields are used per element.)   |

| Descriptor | Meaning   |
|------------|---|
| TOUT       | Type of matrix that will be created: (Integer) <ul style="list-style-type: none"> <li>1 Real, single precision</li> <li>2 Real, double precision</li> <li>3 Complex, single precision</li> <li>4 Complex, double precision</li> </ul> |
| GJ, Gi     | Grid, scalar, RINGFL fluid point, PRESPT pressure point, FREEPT free surface displacement, or extra point identification number. (Integer > 0)  |
| CJ, Ci     | Component number for GJ or Gi grid point ( $0 \leq \text{Integer} \leq 6$ ; Blank or zero if GJ or Gi is a scalar, fluid, or extra point.)  |
| NJ, Ni     | Harmonic number of RINGFL point. Must be blank if a point type other than RINGFL is used. A negative number implies the “sine” series; a positive number implies the “cosine” series. (Integer)                                       |
| Ai, Bi     | Real and imaginary parts of matrix element; row (Gi, Ci, Ni) column (GJ, CJ, NJ). If the matrix is real (TIN = 1), then Bi must be blank.   |

1. This entry is allowed only if an AXIF entry is also present.
2. Matrices defined on this entry may be used in dynamics by selection with the Case Control commands K2PP = NAME, B2PP = NAME, or M2PP = NAME for  $[K_{pp}^2]$ ,  $[B_{pp}^2]$ , or  $[M_{pp}^2]$ , respectively. See [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
3. Field 3 or the header entry must contain an integer 0.
4. For symmetric matrices, either the upper or the lower triangle terms may be specified, but not both.
5. Only nonzero terms need be entered.
6. If any DMIAX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, DMIAX entry changes or additions are not recommended on restart.

**DMIG****Direct Matrix Input at Points**

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

| 1    | 2    | 3   | 4   | 5   | 6    | 7     | 8 | 9    | 10 |
|------|------|-----|-----|-----|------|-------|---|------|----|
| DMIG | NAME | "0" | IFO | TIN | TOUT | POLAR |   | NCOL |    |

**Column Entry Format:**

|      |      |    |    |    |    |        |    |    |  |
|------|------|----|----|----|----|--------|----|----|--|
| DMIG | NAME | GJ | CJ |    | G1 | C1     | A1 | B1 |  |
|      |      | G2 | C2 | A2 | B2 | -etc.- |    |    |  |

**Example:**

|      |      |    |   |        |    |    |      |      |    |
|------|------|----|---|--------|----|----|------|------|----|
| DMIG | STIF | 0  | 1 | 3      | 4  |    |      |      |    |
| DMIG | STIF | 27 | 1 |        | 2  | 3  | 3.+5 | 3.+3 |    |
|      |      | 2  | 4 | 2.5+10 | 0. | 50 |      | 1.0  | 0. |

| Descriptor | Meaning  |
|------------|--|
| NAME       | Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)                        |
| IFO        | Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) |
|            | 1            Square  |
|            | 2 or 9      Rectangular  |
|            | 6            Symmetric   |
| TIN        | Type of matrix being input: (Integer)  |
|            | 1            Real, single precision (One field is used per element.)   |
|            | 2            Real, double precision (One field is used per element.)   |
|            | 3            Complex, single precision (Two fields are used per element.)  |
|            | 4            Complex, double precision (Two fields are used per element.)  |
| TOUT       | Type of matrix that will be created: (Integer)   |
|            | 0            Set by precision system cell (Default)  |
|            | 1            Real, single precision  |
|            | 2            Real, double precision  |
|            | 3            Complex, single precision   |

| Descriptor | Meaning  |
|------------|--|
|            | 4           Complex, double precision  |
| POLAR      | Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)              |
| NCOL       | Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5. and 6. (Integer > 0)                                      |
| GJ         | Grid, scalar or extra point identification number for column index. (Integer > 0)  |
| CJ         | Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)                                |
| Gi         | Grid, scalar, or extra point identification number for row index. (Integer > 0)  |
| Ci         | Component number for Gi for a grid point. (0 < CJ $\leq$ 6; blank or zero if Gi is a scalar or extra point.)                               |
| Ai, Bi     | Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real) |

**Remarks:**

1. Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, B2PP = NAME, M2PP = NAME for  $[K_{pp}]$ ,  $[B_{pp}]$ , or  $[M_{pp}]$ , respectively. Matrices may also be selected for all solution sequences by K2GG = NAME, B2GG = NAME, and M2GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.
2. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but input of an element of the matrix more than once will produce a fatal message.
3. Field 3 of the header entry must contain an integer 0.
4. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.
5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term is used for the column index. The CJ term is ignored.
6. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

- If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.
7. The matrix names must be unique among all DMIGs.
  8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double-field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
  9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.
  10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

**DMIG,UACCEL****Direct Matrix Input of Enforced Static Acceleration**

Defines rigid body accelerations in the basic coordinate system.

**Format:**

| 1    | 2      | 3   | 4   | 5   | 6  | 7  | 8  | 9    | 10 |
|------|--------|-----|-----|-----|----|----|----|------|----|
| DMIG | UACCEL | "0" | "9" | TIN |    |    |    | NCOL |    |
| DMIG | UACCEL | L   |     |     | G1 | C1 | X1 |      |    |
|      | G2     | C2  | X2  |     | G3 | C3 | X3 |      |    |

**Example:**

|      |        |   |   |   |   |   |       |   |  |
|------|--------|---|---|---|---|---|-------|---|--|
| DMIG | UACCEL | 0 | 9 | 1 |   |   |       | 4 |  |
| DMIG | UACCEL | 2 |   |   | 2 | 3 | 386.4 |   |  |
| DMIG | UACCEL | 3 |   |   | 2 | 4 | 3.0   |   |  |
| DMIG | UACCEL | 4 |   |   | 2 | 6 | 1.0   |   |  |

| Descriptor | Meaning   |
|------------|---|
| TIN        | Type of matrix being input. (Integer 1 or 2)  |
| 1          | Real, single precision (One field is used per element.)   |
| 2          | Real, double precision (One field is used per element.)   |
| NCOL       | Number of columns, see Remark 2. Default is the number of columns specified. (Integer $> 0$ )         |
| L          | Load sequence number. (Integer $> 0$ )  |
| Gi         | Grid point identification number of a single reference point. (Integer $> 0$ )                        |
| Ci         | Component number for Gi in the basic coordinate system. See Remark 4. ( $0 < \text{Integer} \leq 6$ ) |
| Xi         | Value of enforced acceleration term in the basic coordinate system. (Real)                            |

**Remarks:**

1. DMIG,UACCEL is an optional entry when PARAM,INREL,-1 is specified in SOLs 101 or 200. If DMIG,UACCEL is present, the loads applied to the structure are the sum of the conventional applied loads plus the inertia loads resulting from the rigid body accelerations defined on this entry. If it is not present, conventional inertia relief calculations are performed.
2. The load sequence number interpretation depends on the value of the NCOL field. The recommended method is to set it equal to the number of loading conditions. The load sequence number L is then the sequence number of the subcase to which the applied acceleration will be applied.

3. The grid point identification number listed on Gi defines a single grid point on the model where loads will be applied to cause the enforced acceleration state. Gi must also appear on a SUPPORT Bulk Data entry. It must also appear on a PARAM,GRDPNT entry. In superelement analysis, it must be a residual structure point exterior to all superelements.
4. The Xi value is the enforced acceleration at grid point Gi. The translation and rotation components are in consistent units and will be applied in the basic coordinate system regardless of the displacement coordinate system specified for Gi (CD field on GRID entry).
5. Only nonzero terms need be entered.
6. See [Superelement Analysis](#) in the *MSC Nastran Reference Guide* for the theoretical basis of inertia relief with superelements.
7. If any DMIG entry is changed or added on restart then a complete re-analysis is performed.  
Therefore, DMIG entry changes or additions are not recommended on restart.

**DMIGOUT****DMIG Matrices to be Output from the Marc Portion of SOL 600**

Defines DMIG matrices to be output from the Marc Portion of SOL 600.

**Header Entry Format:**

| 1       | 2  | 3      | 4     | 5     | 6     | 7     | 8       | 9     | 10 |
|---------|----|--------|-------|-------|-------|-------|---------|-------|----|
| DMIGOUT | ID | ISTIFF | IDIFF | IMASS | IDAMP | ICOND | ISPECIF | ISOL  |    |
|         |    | ICTRL  | IFREQ | ICORD | KIND  | AMIN  | IUSEK   | IUSEM |    |
|         |    | IE1    | THRU  | IE2   |       | IE3   | THRU    | IE4   |    |

**Example:**

|         |     |   |    |   |   |         |   |     |  |
|---------|-----|---|----|---|---|---------|---|-----|--|
| DMIGOUT | 100 | 1 | 1  | 0 | 0 | 0       | 0 | 103 |  |
|         |     | 2 | -1 | 2 | 2 | 1.0E-16 |   |     |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example if the Case Control contains SUBCASE 20, ID would be 20. To output matrices in Marc's phase zero, etc ID=0. (Integer; Default = 1, see Remark 11.) |
| ISTIFF     | Flag to output stiffness matrices. (Normally contains the differential stiffness and all other stiffness terms.) (Integer; Default = 0)   |
|            | 0      Do not output the matrix   |
|            | 1      Output the matrix  |
| IDIFF      | Flag to output differential stiffness matrices. (Only available for buckling analysis or subcases.) (Integer; Default = 0)  |
|            | 0      Do not output the matrix   |
|            | 1      Output the matrix  |
| IMASS      | Flag to output mass matrices. (Integer; Default = 0)  |
|            | 0      Do not output the matrix   |
|            | 1      Output the matrix (must be a dynamic analysis)   |
| IDAMP      | Flag to output damping matrices. (Integer; Default = 0)   |
|            | 0      Do not output the matrix   |
|            | 1      Output the matrix (must be a dynamic analysis)   |
| ICOND      | Flag to output conductivity matrices. (Integer; Default = 0)  |
|            | 0      Do not output the matrix   |
|            | 1      Output the matrix  |
| ISPECIF    | Flag to output specific heat matrices. (Integer; Default = 0)   |

| Descriptor | Meaning  |
|------------|--|
|            | 0      Do not output the matrix<br>1      Output the matrix  |
| ISOL       | Solution sequence to run using the DMIG matrices. (Integer absolute value > 100; Default = 0, which means do not run any solution sequence using the DMIG's created by Marc in this execution). To speed up the solution, use DOMAINsolver ACMS (PARTOPT=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112).              |
| ICTRL      | Controls type of matrix. (Integer; Default = 2)<br><br>1      Element matrices<br>2      Global matrices<br>3      Element and global matrices   |
| IFREQ      | Controls how often the matrices are output (Integer; no Default)<br><br>1      Output at every increment<br>2      Output every other increment, etc.<br>3      Output every 3rd increment, etc.<br>-1     Output only at start of the subcase. (See Remark 12.)   |
| ICORD      | Controls matrix output coordinate system (Integer; no Default)<br><br>1      Nastran basic coordinate system<br>2      Current transformed coordinate system (at the start of the run, this is the Nastran global coordinate system)   |
| KIND       | Controls which elements are written (Integer; Default = 2)<br><br>1      A list of elements starting with the 3 <sup>rd</sup> entry will be specified<br>2      All elements will be written   |
| AMIN       | Values below AMIN will not be written (Real; Default = 1.0E-15). Values below AMIN will be skipped.  |
| IUSEK      | Increment to use in SOL 600 CONTINUE option for the stiffness matrix (if CONTINUE is specified on the SOL 600 entry). (Integer; Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global stiffness matrix for increment 10 will have the name jid.marc_cglsti_0020. |
| IUSEM      | Increment to use in SOL 600 CONTINUE option for the mass matrix (If CONTINUE is specified on the SOL 600 entry). (Integer, Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global mass matrix for increment 20 will have the name jid.marc_cgmas_0020.            |

| Descriptor | Meaning  |
|------------|--|
| IE1, IE3   | Starting element number of a range of elements (Integer; no Default) Only enter IE1 if KIND=1                                    |
| IE2, IE4   | Ending element number of a range of elements (Integer; no Default) Only enter IE2 if KIND=1 (IE2 is required if IE1 is entered). |

**Remarks:**

1. The first continuation line is required.
2. The second continuation line should only be entered if KIND=1 and may be repeated as many times as necessary to define all applicable elements.
3. ICTRL, IFREQ, ICORD, KIND and AMIN apply to all types of matrices to be written for the subcase.
4. Only one DMIGOUT entry can be entered per subcase. If more than one is entered, only the first encountered will be used.
5. DMIGOUT entries may be made for each subcase desired.
6. Marc DMIG element output will be in files named jid.marc\_dmigXX\_inc where XX is shown below and inc is the increment number.

|    |                                 |
|----|---------------------------------|
| ST | Stiffness Matrices              |
| DF | Differential stiffness matrices |
| MS | Mass matrices                   |
| DM | Damping                         |
| CO | Conductivity matrices           |
| SP | Specific heat matrices          |

Global matrices are named jid\_marc\_cglsti\_inc.

7. If the SOL 600 CONTINUE options is invoked, Case Control commands and a Bulk Data include statement to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
8. ID may not be 600 or 700 in the Executive Control statement, SOL 600, ID.
9. The following Bulk Data parameters are usually required in addition to the DMIGOUT entry:

\$2345678x234567890123456x34567890123456

param\* mrspawn2 nastcmd  
param,mrmtxnam,KAAK  
param,marcfile,nastb.rc

where



- a. nastcmd is the name of the command to run the primary and continuation jobs (examples are nastran, nast2006t1, nast2006t2, etc.)
  - b. nastb.rc should be changed to the name of the rc file to be used for the continuation run. It usually will specify mem= with a larger value than that of the primary run and also include a line bat=no (except for windows systems).
  - c. PARAM,MARCFILi should not be used.
10. For standard nonlinear static or dynamic analyses the stiffness matrix contains all contributions including the differential stiffness matrix and it is not possible to obtain the differential stiffness matrix separately. For a buckling analysis, the differential stiffness matrix may be obtained separately.
11. To obtain stiffness and mass matrices for SOL 600,103, ID in field 2 of this entry must be zero.
12. If IFREQ is 1 or a small number, the number of matrices output for dynamic analyses (SOL 600,109 or SOL 600,120) can be extremely large.
13. If ID=0, IFREQ should be 1.
14. Setting IFREQ to a value larger than the actual number of increments in a subcase will produce no matrices.

**DMIGROT****DMIG Matrices for which large rotation in SOL 600**

Defines large rotation and other characteristics of a matrix entered using DMIG in SOL 600.

Format for K2GG and K2PP Stiffness DMIG Matrices:

| 1       | 2    | 3  | 4  | 5  | 6     | 7     | 8    | 9      | 10 |
|---------|------|----|----|----|-------|-------|------|--------|----|
| DMIGROT | Name | G1 | G2 | G3 | SCALE | ITRAN | IACT | IDEACT |    |

Format for M2GG and M2PP Mass DMIG Matrices:

|         |      |  |  |  |       |  |      |  |  |
|---------|------|--|--|--|-------|--|------|--|--|
| DMIGROT | Name |  |  |  | SCALE |  | IACT |  |  |
|---------|------|--|--|--|-------|--|------|--|--|

Example:

|         |      |     |     |     |       |   |   |   |  |
|---------|------|-----|-----|-----|-------|---|---|---|--|
| DMIGROT | KAAX | 375 |     |     |       |   |   |   |  |
| DMIGROT | KAAY | 401 | 402 | 403 |       |   |   |   |  |
| DMIGROT | KAAX |     |     |     | 1.25  | 1 | 2 | 5 |  |
| DMIGROT | MAAX |     |     |     | 0.001 |   | 7 | 8 |  |

| Descriptor | Meaning  |
|------------|--|
| Name       | Name of the DMIG matrix for which large rotation is specified. See Remark 1. (Character; Required field; no Default)   |
| G1         | Grid ID specifying the rotation of the matrix. See Remark 2. (Integer; Required field; no Default)   |
| G2         | Grid ID which combined with G1 and G3 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0; )  |
| G3         | Grid ID which combined with G1 and G2 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0)  |
| SCALE      | Scale factor. Each term in the DMIG matrix will be scaled by this value. (Real; Default = 1.0)   |
| ITRAN      | Flag indicating whether to transform the DMIG's or not. (Integer, Default = 0) <ul style="list-style-type: none"> <li>0      Do not transform the DMIG matrix.</li> <li>1      Transform the DMIG using G1, G2, G3 to form the transformation matrix.</li> </ul> |
| IACT       | Subcase ID for which this DMIG matrix will be activated. See Remark 5. (Integer; Default = 0). <ul style="list-style-type: none"> <li>0      The matrix is active for all subcases.</li> <li>N      The matrix is activated starting with subcase N</li> </ul>   |
| IDEACT     | Subcase ID for which this DMIG matrix will be deactivated. See Remark 5. (Integer; no Default; leave blank if the matrix should be active fro the entire analysis or starting with subcase IACT)   |

Remarks:

1. All DMIG matrices with the name specified in field 2 will rotate with grids G1.
2. If G1 has 6 dof it is not necessary to specify G2 or G3 and the DMIG matrix will rotate with the same rotation as G1. If G1 has fewer than 6 dof, or if it is desired that the DMIG matrix rotate with a plane defined by G1, G2 and G3, G2 and G3 must be entered and may not all be collinear.
3. If a transformation is to be applied to the stiffness matrix, the DMIG **must** contain all of the degrees of freedom associated with the node to which the transformation is applied.
4. This bulk data entry maps to Marc's K2GG (and K2PP) model definition entry.
5. IACT and IDEACT are ID's of the Case Control SUBCASE entry.
6. G1, G2, G3 and ITRAN may not be entered for mass matrices.

**DMIJ****Direct Matrix Input at js-Set of the Aerodynamic Mesh**

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the interference elements of a CAERO2, use DMIJI or DMI.

**Header Entry Format:**

| 1    | 2    | 3   | 4   | 5   | 6    | 7     | 8 | 9    | 10 |
|------|------|-----|-----|-----|------|-------|---|------|----|
| DMIJ | NAME | "0" | IFO | TIN | TOUT | POLAR |   | NCOL |    |

**Column Entry Format:**

|      |      |    |    |    |    |        |    |    |  |
|------|------|----|----|----|----|--------|----|----|--|
| DMIJ | NAME | GJ | CJ |    | G1 | C1     | A1 | B1 |  |
|      |      | G2 | C2 | A2 | B2 | -etc.- |    |    |  |

**Example:**

|      |       |   |   |    |   |   |    |   |  |
|------|-------|---|---|----|---|---|----|---|--|
| DMIJ | ALPH1 | 0 | 9 | 2  | 0 |   |    | 1 |  |
| DMIJ | ALPH1 | 1 | 1 |    | 1 | 1 | .1 |   |  |
|      |       | 2 | 1 | .1 |   |   |    |   |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.) |
| IFO        | Form of matrix being input. (Integer)   |
|            | 1            Square   |
|            | 9 or 2      Rectangular   |
|            | 6            Symmetric  |
| TIN        | Type of matrix being input: (Integer)   |
|            | 1            Real, single precision (One field is used per element)   |
|            | 2            Real, double precision (One field is used per element)   |
|            | 3            Complex, single precision (Two fields are used per element)                                    |
|            | 4            Complex, double precision (Two fields are used per element)                                    |
| TOUT       | Type of matrix being created: (Integer)   |
|            | 0            Set by precision system cell (Default)   |
|            | 1            Real, single precision   |

| Descriptor | Meaning  |
|------------|--|
| 2          | Real, double precision   |
| 3          | Complex, single precision  |
| 4          | Complex, double precision  |
| POLAR      | Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)            |
| NCOL       | Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)  |
| GJ         | Grid, scalar or extra point identification number for column index. (Integer > 0)  |
| CJ         | Component number for grid point GJ. ( $0 < \text{Integer} \leq 6$ ; blank or zero if GJ is a scalar or extra point.)                       |
| Gi         | Grid, scalar, or extra point identification number for row index. (Integer > 0)  |
| Ci         | Component number for Gi for a grid point. ( $0 < CJ \leq 6$ ; blank or zero if Gi is a scalar or extra point.)                             |
| Ai, Bi     | Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real) |

**Remarks:**

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. Also, DMIJ may also be used for the W2GJ and FA2J entries. Again, a single column is required. If both DMI and DMIJ are specified for W2GJ or FA2J, the DMI entry will be used. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJ data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

| Entry Type | COMP |   |   |   |   |
|------------|------|---|---|---|---|
|            | 1    | 2 | 3 | 5 | 6 |
| CAERO1     |      |   | X |   |   |
| CAERO2-Y   |      | X |   |   |   |
| CAERO2-Z   |      |   | X |   |   |
| CAERO2-ZY  |      | X | X |   |   |

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all DMIJ matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJ.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJ input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

## DMIJI

## Direct Matrix Input at js-Set of the Interference Body

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the slender elements of a CAERO2, or a CAERO1, 3, 4 or 5 use DMIJ or DMI.

**Header Entry Format:**

| 1     | 2    | 3   | 4   | 5   | 6    | 7     | 8 | 9    | 10 |
|-------|------|-----|-----|-----|------|-------|---|------|----|
| DMIJI | NAME | "0" | IFO | TIN | TOUT | POLAR |   | NCOL |    |

**Column Entry Format:**

|      |      |    |    |    |    |        |    |    |  |
|------|------|----|----|----|----|--------|----|----|--|
| DMIJ | NAME | GJ | CJ |    | G1 | C1     | A1 | B1 |  |
|      |      | G2 | C2 | A2 | B2 | -etc.- |    |    |  |

**Example:**

|       |       |   |   |    |   |   |    |   |  |
|-------|-------|---|---|----|---|---|----|---|--|
| DMIJI | ALPH1 | 0 | 9 | 2  | 0 |   |    | 1 |  |
| DMIJI | ALPH1 | 1 | 1 |    | 1 | 1 | .1 |   |  |
|       |       | 2 | 1 | .1 |   |   |    |   |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.) |
| IFO        | Form of matrix being input. (Integer)   |
|            | 1            Square   |
|            | 9 or 2      Rectangular   |
|            | 6            Symmetric  |
| TIN        | Type of matrix being input: (Integer)   |
|            | 1            Real, single precision (One field is used per element)   |
|            | 2            Real, double precision (One field is used per element)   |
|            | 3            Complex, single precision (Two fields are used per element)                                    |
|            | 4            Complex, double precision (Two fields are used per element)                                    |
| TOUT       | Type of matrix being created: (Integer)   |
|            | 0            Set by precision system cell (Default)   |
|            | 1            Real, single precision   |
|            | 2            Real, double precision   |

| Descriptor | Meaning  |
|------------|--|
| 3          | Complex, single precision  |
| 4          | Complex, double precision  |
| POLAR      | Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)            |
| NCOL       | Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)  |
| GJ         | Grid, scalar or extra point identification number for column index. (Integer > 0)  |
| CJ         | Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)                                     |
| Gi         | Grid, scalar, or extra point identification number for row index. (Integer > 0)  |
| Ci         | Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)                                    |
| Ai, Bi     | Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real) |

#### Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJI data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

| Entry Type | COMP |   |   |   |   |
|------------|------|---|---|---|---|
|            | 1    | 2 | 3 | 5 | 6 |
| CAERO2-Y   |      | X |   |   |   |
| CAERO2-Z   |      |   | X |   |   |
| CAERO2-ZY  |      | X | X |   |   |

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.

5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all DMIJI matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJI.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJI input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

**DMIK****Direct Matrix Input at ks-Set of the Aerodynamic Mesh**

Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points. These include WKK, WTFAC and input forces associated with AEFORCE entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

| 1    | 2    | 3   | 4   | 5   | 6    | 7     | 8 | 9    | 10 |
|------|------|-----|-----|-----|------|-------|---|------|----|
| DMIK | NAME | "0" | IFO | TIN | TOUT | POLAR |   | NCOL |    |

**Column Entry Format:**

|      |      |    |    |    |    |        |    |    |  |
|------|------|----|----|----|----|--------|----|----|--|
| DMIK | NAME | GJ | CJ |    | G1 | C1     | A1 | B1 |  |
|      |      | G2 | C2 | A2 | B2 | -etc.- |    |    |  |

**Example:**

|      |       |   |   |     |   |   |     |   |  |
|------|-------|---|---|-----|---|---|-----|---|--|
| DMIK | ALPH1 | 0 | 9 | 2   | 0 |   |     | 1 |  |
| DMIK | ALPH1 | 1 | 1 |     | 1 | 1 | 1.0 |   |  |
|      |       | 2 | 1 | 1.0 |   |   |     |   |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.) |
| IFO        | Form of matrix being input. (Integer)   |
|            | 1        Square   |
|            | 9 or 2    Rectangular   |
|            | 6        Symmetric  |
| TIN        | Type of matrix being input: (Integer)   |
|            | 1        Real, single precision (One field is used per element)   |
|            | 2        Real, double precision (One field is used per element)   |
|            | 3        Complex, single precision (Two fields are used per element)  |
|            | 4        Complex, double precision (Two fields are used per element)  |
| TOUT       | Type of matrix being created: (Integer)   |
|            | 0        Set by precision system cell (Default)   |
|            | 1        Real, single precision   |
|            | 2        Real, double precision   |
|            | 3        Complex, single precision  |

| Descriptor | Meaning  |
|------------|--|
| 4          | Complex, double precision  |
| POLAR      | Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)            |
| NCOL       | Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)  |
| GJ         | Grid, scalar or extra point identification number for column index. (Integer > 0)  |
| CJ         | Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)                                |
| Gi         | Grid, scalar, or extra point identification number for row index. (Integer > 0)  |
| Ci         | Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)                                    |
| Ai, Bi     | Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real) |

**Remarks:**

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. If both DMI and DMIK are specified for WKK or WTFACT, the DMI entry will be used. DMI may NOT be used for AEFORCE.
2. The ks-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIK data will be partitioned to the k-set, not reduced. No warnings are issued about truncated data. The k-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

| Entry Type | COMP |   |   |   |   |
|------------|------|---|---|---|---|
|            | 1    | 2 | 3 | 5 | 6 |
| CAERO1     |      |   | X | X |   |
| CAERO2-Y   |      | X |   | X |   |
| CAERO2-Z   |      |   | X |   | X |
| CAERO2-ZY  |      | X | X | X | X |

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIK matrices is always the ks-set size--the union of the k-set and the permanently SPC'd partition). The GJ term is used for the column index. The CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIK.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIK matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIK input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIK entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIK entry changes or additions are not recommended on restart.

**DOPTPRM****Design Optimization Parameters**

Overrides default values of parameters used in design optimization.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| DOPTPRM | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 | PARAM4 | VAL4 |    |
|         | PARAM5 | VAL5 | -etc.- |      |        |      |        |      |    |

**Example:**

|         |        |   |        |    |  |  |  |  |  |
|---------|--------|---|--------|----|--|--|--|--|--|
| DOPTPRM | IPRINT | 5 | DESMAX | 10 |  |  |  |  |  |
|---------|--------|---|--------|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PARAMi     | Name of the design optimization parameter. Allowable names are given in <a href="#">Table 11</a> . (Character) |
| VALi       | Value of the parameter. (Real or Integer, see <a href="#">Table 11</a> .)                                      |

**Table 11**      **PARAMi Names and Descriptions**

| Name   | Description, Type, and Default Value   |   |
|--------|--|---|
| APRCOD | Approximation method to be used. (Integer 1, 2, or 3; Default = 2)   |   |
|        | 1  | Direct Linearization. APRCOD = 1 is recommended for shape, topology and topography optimization problems. |
|        | 2  | Mixed Method based on response type   |
|        | 3  | Convex Linearization  |
| AUTOSE | Flag to request an AESO job. AUTOSE = 1 activates an AESO creation run. (Integer $\geq 0$ ; Default = 0)   |   |
| CONV1  | Relative criterion to detect convergence. If the relative change in objective between two optimization cycles is less than CONV1, then optimization is terminated. Topology optimization can be terminated with CONV1<5.00E-3 (Real > 0.0; Default = 0.001). |   |
| CONV2  | Absolute criterion to detect convergence. If the absolute change in objective between two optimization cycles is less than CONV2, then optimization is terminated. (Real > 0.0; Default = 1.0E-20)   |   |
| CONVDV | Relative convergence criterion on design variables. (Real > 0.0; Default = 0.001 for non-topology; Default = 0.0001 for topology optimization)   |   |
| CONVPR | Relative convergence criterion on properties. (Real > 0.0; Default = 0.001)  |   |

Table 11 PARAMi Names and Descriptions (continued)

| Name    | Description, Type, and Default Value  |  |
|---------|---|--|
| CT      | Constraint tolerance. Constraint is considered active if current value is greater than CT. (Real < 0.0; Default = -0.03)  |  |
| CTMIN   | Constraint is considered violated if current value is greater than CMIN. (Real > 0.0; Default = 0.003)  |  |
| DABOBJ  | Maximum absolute change in objective between two consecutive iterations to indicate convergence at optimizer level. F0 is the initial objective function value. (Real > 0.0; Default = MAX[0.001*ABS(F0), 0.0001])    |  |
| DELB    | Relative finite difference move parameter. (Real > 0.0; Default = 0.0001; 0.001 for fatigue responses)  |  |
| DELOBJ  | Maximum relative change in objective between two consecutive iterations to indicate convergence at optimizer level. (Real > 0.0; Default = 0.001)   |  |
| DELP    | Fractional change allowed in each property during any optimization design cycle. This provides constraints on property moves. (Real, 0.0<DELP<1.0, default=0.2)   |  |
| DELX    | Fractional change allowed in each design variable during any optimization cycle. (Real > 0.0; Default = .5 for sizing/shape/topometry optimization; Default = 0.2 for topology and topography optimization)           |  |
| DELXESL | Fractional change allowed in each design variable during the ESLNRO loop. (Real > 0.0, Default = 0.5)   |  |
| DESMAX  | Maximum number of design cycles (not including FSD cycle) to be performed. (Integer $\geq 0$ ; Default = 5 for sizing/shape optimization; Default = 30 for topology, topography, and topometry optimization)          |  |
| DISCOD  | Discrete Processing Method: (Integer 0, 1, 2, 3 or 4; Default = 1)  |  |
|         | 0   | No Discrete Optimization                 |
|         | 1   | Design of Experiments                    |
|         | 2   | Conservative Discrete Design             |
|         | 3   | Round up to the nearest design variable  |
|         | 4   | Round off to the nearest design variable |
| DISBEG  | Design cycle ID for discrete variable processing initiation. Discrete variable processing analysis is carried out for every design cycle after DISBEG. (Integer $\geq 0$ ; Default = 0=the last design cycle)         |  |
| DPMAX   | Maximum fraction of change on designed property (Default = 0.5), used by Trust Region Method.   |  |
| DPMIN   | Minimum move limit imposed. (Real > 0.0; Default = 0.01)  |  |
| DRATIO  | Threshold value that can be used to turn off an active AESO job. An AESO job is terminated if the ratio of the size of a design model to that of an analysis model is greater than DRATIO. (Real > 0.; Default = 0.1) |  |

Table 11 PARAMi Names and Descriptions (continued)

| Name              | Description, Type, and Default Value   |   |
|-------------------|--|---|
| DSMXESL           | Maximum number of design cycles applied to the ESLNRO loop. (Integer $\geq 0$ ; Default = 20)  |   |
| DXMAX             | Maximum fraction of change on design variable (Default = 1.0), used by Trust Region Method.  |   |
| DXMIN             | Minimum design variable move limit. (Real $> 0.0$ ; Default = 0.05 for sizing/shape/topometry optimization; Default = 1.0E-5 for topology and topography optimization)   |   |
| ETA1 ( $\eta_1$ ) | the cutting ratio 1 (Default = 0.01), used by Trust Region Method.   |   |
| ETA2 ( $\eta_2$ ) | the cutting ratio 2 (Default = 0.25), used by Trust Region Method.   |   |
| ETA3 ( $\eta_3$ ) | the cutting ratio 3 (Default = 0.7), used by Trust Region Method.  |   |
| FSDALP            | Relaxation parameter applied in Fully Stressed Design. (Real, $0.0 < FSDALP \leq 1.0$ ; Default = 0.9)   |   |
| FSDMAX            | Specifies the number of Fully Stressed Design Cycles that are to be performed. (Integer; Default = 0)  |   |
| GMAX              | Maximum constraint violation allowed at the converged optimum. (Real $> 0.0$ ; Default = 0.005)  |   |
| GSCAL             | Constraint normalization factor. See Remarks under the DSCREEN and DCONSTR entries. (Real $> 0.0$ ; Default = 0.001)   |   |
| IGMAX             | If IGMAX = 0, only gradients of active and violated constraints are calculated. If IGMAX $> 0$ , up to IGMAX gradients are calculated including active, violated, and near active constraints. For many constraint problems, a reasonable value of IGMAX = 2*NDV (number of design variables). A smaller IGMAX has a faster performance of MSCADS. (Integer $> 0$ ; Default = 0) |   |
| IPRINT            | Print control during approximate optimization phase. Increasing values represent increasing levels of optimizer information. ( $0 \leq \text{Integer} \leq 7$ ; Default = 0)   |   |
|                   | 0  | No output (Default)   |
|                   | 1  | Internal optimization parameters, initial information, and results      |
|                   | 2  | Same, plus objective function and design variables at each iterations   |
|                   | 3  | Same, plus constraint values and identification of critical constraints |
|                   | 4  | Same, plus gradients  |
|                   | 5  | Same, plus search direction   |
|                   | 6  | Same, plus scaling factors and miscellaneous search information         |
|                   | 7  | Same, plus one dimensional search information                           |
| ISCAL             | Design variables scaling. (Integer $\geq -1$ ; Default = 0) = -1 no scaling; $\geq 0$ scaling  |   |

Table 11 PARAMi Names and Descriptions (continued)

| Name   | Description, Type, and Default Value   |   |
|--------|--|---|
| METHOD | Optimization Method: (Integer $\geq 0$ ; Default = 0)  |   |
|        | 0  | Automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory. |
|        | 1  | Modified Method of Feasible Directions for MSCADS.  |
|        | 2  | Sequential Linear Programming for MSCADS  |
|        | 3  | Sequential Quadratic Programming for MSCADS   |
|        | 4  | SUMT method for MSCADS  |
| NASPR0 | IJK  | See Remark 3.   |
|        | First cycle analysis output control. (Integer 0 or 1)  |   |
|        | 0  | Print analysis output of first cycle. (Default)   |
| OBJMOD | 1  | Do NOT print analysis output of first cycle.  |
|        | Objective function modification. (Integer; Default = 0)  |   |
|        | 0  | Objective function will not be modified.  |
| OPTCOD | 1  | Objective function will be reset to 0.0. Subsequently, printed objective function value represents the change of objective function.                                |
|        | OPTCOD. See Remark 2. (Character; Default = Blank)   |   |
|        | Blank  | Taken from system cell number 413   |
|        | “MSCADS”   | MSCADS is used  |
|        | “IPOPT”  | IPOPT is used   |
|        | “External Optimizer SCA Service Identifier”  |   |
| P1     |  |   |
|        | Must be defined in the CONNECT Service statement (see <i>MSC Nastran User Defined Services User's Guide</i> )  |   |
|        | Print control items specified for P2. (Integer $\geq 0$ ; Default = 0) Initial results are always printed prior to the first approximate optimization. If an optimization task is performed, final results are always printed for the final analysis unless PARAM,SOFTEXIT,YES is specified. These two sets of print are not controllable. |   |
|        |  | n Print at every n-th design cycle.   |

Table 11 PARAMi Names and Descriptions (continued)

| Name   | Description, Type, and Default Value  |  |
|--------|---|--|
| P2     | Items to be printed according to P1: (Integer; Default = 1)   |  |
| 0      | No print.   |  |
| 1      | Print objective and design variables. (Default for sizing/shape optimization)<br>Print objective. (Default for topology optimization)<br>$P2 \geq 13$ Print design variables for topology, topography, and topometry optimization   |  |
| 2      | Print properties.   |  |
| 4      | Print constraints.  |  |
| 8      | Print responses.  |  |
| 16     | Print weight as a function of a material ID (note that this is not a design quantity so that only inputs to the approximate design are available).  |  |
| n      | Sum of desired items. For example, P2 = 10 means print properties and responses.  |  |
| P2CALL | Maximum number of retained constraints of all categories to be printed per category. This single parameter can be used in place of the individual parameters P2CBL, P2CC, P2CDDV, P2CM, P2CP and P2CR. If any of these six parameters are non-zero, the P2CALL value is overridden for that constraint type. (Integer $> 0$ ; default is to print all retained constraints.)  |  |
| P2CBL  | Maximum number of constraints on beam library dimensions to be printed. (Integer $\geq 0$ ; default is to print all beam library constraints.)  |  |
| P2CC   | Maximum number of constraints on connectivity properties to be printed. (Integer $\geq 0$ ; default is to print all connectivity property constraints.)   |  |
| P2CDDV | Maximum number of constraints on dependent design variables to be printed. (Integer $\geq 0$ ; default is to print all dependent design variable constraints.)  |  |
| P2CM   | Maximum number of constraints on material properties to be printed. (Integer $\geq 0$ ; default is to print all material property constraints.)   |  |
| P2CP   | Maximum number of constraints on element properties to be printed. (Integer $\geq 0$ ; default is to print all element property constraints.)   |  |
| P2CR   | Maximum number of constraints on design responses to be printed. (Integer $\geq 0$ ; default is to print all retained design response constraints.)   |  |
| P2RSET | ID of a SET1 Bulk Data entry to identify the set of retained responses (DRESP1, DRESP2 and/or DRESP3) to be printed. (Integer; Default is to print all responses associated with printed constraints. If P2CR is $> 0$ , the set associated P2RSET $> 0$ will be printed independent of the responses associated with the printed constraint. If P2CR $> 0$ and PR2SET = -1, all retained responses will be printed.) |  |

Table 11 PARAMi Names and Descriptions (continued)

| Name    | Description, Type, and Default Value  |  |
|---------|---|--|
| PENAL   | Penalty parameter used to transform an infeasible approximate optimization task to a feasible one. Setting this parameter to; e.g., 2.0 may improve optimizer performance when the starting design is infeasible. (Real; Default = 0.0)   |  |
| PLVIOL  | Flag for handling of property limit violation. By default, the job will terminate with a user fatal message if the property derived from design model (DVPRELi, DVMRELi, DVCRELi) exceeds the property limits. Setting PLVIOL to a non-zero number will cause the program to issue a user warning message by ignoring the property limits violation and proceed with the analysis. (Integer; Default = 0) |  |
| PTOL    | Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry (through DVPRELi relations). PTOL is provided to trap ill-posed design models. (The minimum tolerance may be specified on user parameter DPEPS. See <a href="#">Parameters, 783</a> ) (Real > 0.0; Default = 1.0E+35)     |  |
| STPSCL  | Scaling factor for shape finite difference step sizes, to be applied to all shape design variables. (Real > 0.0; Default = 1.0)   |  |
| TCHECK  | Topology Checkerboarding/minimum member size control option. (Integer $\geq -1$ )   |  |
|         | 1   | Filtering algorithm  |
|         | 2   | Density constraint   |
|         | 0   | No control   |
|         | -1  | Automatic selection of filtering or density constraint algorithm for a better performance. (Default) |
| TDMIN   | Minimum diameter of members in topology optimization. This option is applied on 2 and 3D elements only. (Real > 0.0)  |  |
| TREGION | Flag to invoke Trust Region method.   |  |
|         | 0   | Don't employ trust regions (Default)   |
|         | 1   | Turn Trust Region on   |
| UPDFAC1 | Updating factor 1 (Default = 2.0), used by Trust Region Method.   |  |
| UPDFAC2 | Updating factor 2 (Default = 0.5), used by Trust Region Method.   |  |

**Remarks:**

- Only one DOPTPRM entry is allowed in the Bulk Data Section. All defaults recommended.
- OPTCOD specifies which optimization code to be used in SOL 200 and METHOD specifies which optimization method to be used. The default is recommended.
- METHOD = IJK enables a user selectable optimization strategy as documented in Vanderplaats, G. N., ADS -- A Fortran Program for Automated Design Synthesis -- Version 1.10, NASA CR 177985, 1985. The default is recommended.

The I selects one of ten available strategy options:

|   |   |
|---|---|
| 0 | None -- Go directly to the optimizer  |
| 1 | Sequential unconstrained minimization using the exterior penalty function method                    |
| 2 | Sequential unconstrained minimization using the linear extended interior penalty function method    |
| 3 | Sequential unconstrained minimization using the quadratic extended interior penalty function method |
| 4 | Sequential unconstrained minimization using the cubic extended interior penalty function method     |
| 5 | Augmented Lagrange multiplier method  |
| 6 | Sequential linear programming   |
| 7 | Method of centers   |
| 8 | Sequential quadratic programming  |
| 9 | Sequential convex programming   |

The J selects one of five available optimizer options:

|   |   |
|---|---|
| 1 | Fletcher-Reeves algorithm for unconstrained minimization                                      |
| 2 | Davidon-Fletcher-Powell (DFP) variable metric method for unconstrained minimization           |
| 3 | Broydon-Fletcher-Goldfarb-Shanno (BFGS) variable metric method for unconstrained minimization |
| 4 | Method of feasible directions for constrained minimization                                    |
| 5 | Modified method of feasible directions for constrained minimization                           |

And K selects one of eight available one-dimensional search strategies:

|   |   |
|---|---|
| 1 | Find the minimum of an unconstrained function using the Golden Section method   |
| 2 | Find the minimum of an unconstrained function using the Golden Section method followed by polynomial interpolation                                  |
| 3 | Find the minimum of an unconstrained function by first finding bounds and then using the Golden Section method followed by polynomial interpolation |
| 4 | Find the minimum of an unconstrained function by polynomial interpolation/extrapolation without first finding bounds on the solution                |

|   |   |
|---|---|
| 5 | Find the minimum of a constrained function using the Golden Section method  |
| 6 | Find the minimum of a constrained function using the Golden Section method followed by polynomial interpolation                   |
| 7 | Find the minimum of a constrained function by first finding bounds and then using polynomial interpolation                        |
| 8 | Find the minimum of a constrained function by polynomial interpolation/extrapolation without first finding bounds on the solution |

**DPHASE****Dynamic Load Phase Lead**

Defines the phase lead term  $\theta$  in the equation of the dynamic loading function.

**Format:**

| 1      | 2   | 3  | 4  | 5   | 6  | 7  | 8   | 9 | 10 |
|--------|-----|----|----|-----|----|----|-----|---|----|
| DPHASE | SID | P1 | C1 | TH1 | P2 | C2 | TH2 |   |    |

**Example:**

|        |   |    |   |     |   |   |     |  |  |
|--------|---|----|---|-----|---|---|-----|--|--|
| DPHASE | 4 | 21 | 6 | 2.1 | 8 | 6 | 7.2 |  |  |
|--------|---|----|---|-----|---|---|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of DPHASE entry. (Integer > 0)   |
| Pi         | Grid, extra, or scalar point identification number. (Integer > 0)                                  |
| Ci         | Component number. (Integers 1 through 6 for grid points; zero or blank for extra or scalar points) |
| THi        | Phase lead $\theta$ in degrees. (Real)   |

**Remarks:**

1. One or two dynamic load phase lead terms may be defined on a single entry.
2. SID must be referenced on a RLOADi entry. Refer to the RLOAD1 or RLOAD2 entry for the formulas that define how the phase lead  $\theta$  is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DPHASE entries may only be applied to loads on points in the residual structure.
5. RC network solver does not support DPHASE for thermal analysis.

**DRESP1****Design Sensitivity Response Quantities**

Defines a set of structural responses that is used in the design either as constraints or as an objective.

**Format:**

| 1      | 2    | 3      | 4     | 5     | 6      | 7    | 8    | 9    | 10 |
|--------|------|--------|-------|-------|--------|------|------|------|----|
| DRESP1 | ID   | LABEL  | RTYPE | PTYPE | REGION | ATTA | ATTB | ATT1 |    |
|        | ATT2 | -etc.- |       |       |        |      |      |      |    |

**Example:**

|        |     |     |        |      |   |   |  |     |  |
|--------|-----|-----|--------|------|---|---|--|-----|--|
| DRESP1 | 1   | DX1 | STRESS | PROD | 2 | 3 |  | 102 |  |
|        | 103 |     |        |      |   |   |  |     |  |

|        |    |      |       |        |  |    |  |    |  |
|--------|----|------|-------|--------|--|----|--|----|--|
| DRESP1 | 10 | BMWW | WMPID | PSHELL |  | 10 |  | 10 |  |
|        | 20 | 30   |       |        |  |    |  |    |  |

| Descriptor          | Meaning  |
|---------------------|--|
| ID                  | Unique entry identifier. (Integer > 0)   |
| LABEL               | User-defined label. (Character, no default)  |
| RTYPE               | Response type. See <a href="#">Table 12</a> . (Character)  |
| PTYPE               | Element flag (PTYPE = "ELEM") or property entry name. Used with element type responses (stress, strain, force, etc.) to identify the property type, since property entry IDs are not unique across property types. (Character: "ELEM", "PBAR", "PSHELL", etc.) |
| REGION              | Region identifier for constraint screening. See Remark 10. for defaults. (Integer > 0)   |
| ATTA, ATTB,<br>ATTi | Response attributes. See <a href="#">Table 12</a> . (Integer > 0 or Real or blank)   |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE)                        | Response Attributes  |   |  |
|--|--|---|--|
|  | ATT A (Integer > 0)  | ATT B (Integer > 0 or Real > 0.0)               | ATT I (Integer > 0)                        |
| WEIGHT                                       | Row Number<br>( $1 \leq \text{ROW} \leq 6$ )<br>See Remark 24. | Column Number<br>( $1 \leq \text{COL} \leq 6$ ) | SEIDi or All or blank. See Remark 12.      |
| VOLUME                                       | Blank  | Blank   | SEIDi or ALL or blank. See Remark 12.      |
| FRMASS                                       | Blank  | Blank   | Blank or Property ID (PID). See Remark 37. |
| COMP   | Blank  | Blank   | Blank                                      |
| See Remark 28.                               |  |   |  |
| EIGN   | Normal Modes Mode Number. See Remark 33.                       | Approximation Code. See Remark 19.              | Blank                                      |
| CEIG   | Complex Eigenvalue Mode Number. (Integer > 0)                  | ALPHA or OMEGA (Default = ALPHA)                | Blank                                      |
| FREQ   | Normal Modes Mode Number.<br>See Remarks 18. and 33.           | Approximation Code. See Remark 19.              | Blank                                      |
| LAMA   | Buckling Mode Number   | Approximation Code. See Remark 19.              | Blank                                      |
| DISP   | Displacement Component   | Blank or Mode Number                            | Grid ID                                    |
| STRAIN                                       | Strain Item Code   | Blank or Mode Number                            | Property ID (PID) or Element ID (EID)      |
| ESE  | Strain Energy Item Code<br>See Remark 21.                      | Blank or Mode Number                            | Property ID (PID) or Element ID (EID)      |
| STRESS                                       | Stress Item Code   | Blank or Mode Number                            | Property ID (PID) or Element ID (EID)      |
| FORCE  | Force Item Code  | Blank or Mode Number                            | Property ID (PID) or Element ID (EID)      |
| FATIGUE<br>(pseudo-static)<br>See Remark 39. | Fatigue Item Code.<br>See Remark 43.                           | ID of a FATIGUE case control.<br>See Remark 40. | Property ID (PID) or Element ID (EID)      |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE)  | Response Attributes                              |  |  |
|--|--|--|--|
|  | ATTA (Integer > 0)                               | ATTB (Integer > 0 or Real > 0.0)   | ATTI (Integer > 0)                                     |
| FRFTG<br>(frequency response - random vibration fatigue)<br>See Remark 39. | Fatigue Item Code.<br>See Remark 43.             | ID of a FATIGUE case control.<br>See Remark 40.                                      | Property ID (PID) or Element ID (EID)                  |
| SPCFORCE   | SPC Force Component                              | Blank  | Grid ID  |
| CSTRAIN<br>See Remark 2.   | Strain Item Code                                 | LAMINA Number or GPLYIDi<br>(Integer; Default = 1)                                   | Property ID (PID) or Element ID (EID)                  |
| CSTRESS<br>See Remark 2.   | Stress Item Code                                 | LAMINA Number or GPLYIDi<br>(Integer; Default = 1)                                   | Property ID (PID) or Element ID (EID)                  |
| CFailure<br>See Remark 2.  | Failure Indices Item Code                        | LAMINA Number or GPLYIDi<br>(Integer; Default = 1)                                   | Property ID (PID) or Element ID (EID)                  |
| CSTRAT<br>See Remark 2.  | Composite Strength Ratio Item Code               | LAMINA Number or GPLYIDi<br>(Integer; Default = 1)                                   | Property ID (PID) or Element ID (EID)                  |
| TOTSE<br>(Total Strain Energy)   | Blank  | Blank or Mode Number   | SEIDi or All or blank. See Remark 12.                  |
| GPFORCE  | GPFORCE Component Code (1-6; see Remark 25.)     | Blank  | Element ID   |
| GPFORCP  | Grid Point (see Remark 26.)                      | Blank  | Grid ID connected to ATTA grid to specify orientation. |
| ABSTRESS   | Arbitrary Beam Stress Item Code (see Remark 30.) | Blank  | Property ID (PID) or Element ID (EID)                  |
| FRDISP   | Displacement Component                           | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Grid ID  |
| PRES   | Acoustic Pressure Component (= 1 or 7)           | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Grid ID  |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE) | Response Attributes                                 |  |                                       |
|-----------------------|---|--|---------------------------------------|
|                       | ATT A (Integer > 0)                                 | ATT B (Integer > 0 or Real > 0.0)  | ATT I (Integer > 0)                   |
| FRVELO                | Velocity Component                                  | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Grid ID                               |
| FRACCL                | Acceleration Component                              | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Grid ID                               |
| FRSPCF                | SPC Force Component                                 | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Grid ID                               |
| FRSTRE                | Stress Item Code                                    | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Property ID (PID) or Element ID (EID) |
| FRFORC                | Force Item Code                                     | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Property ID (PID) or Element ID (EID) |
| PSDDISP               | Displacement Component<br>(see Remarks 27. and 31.) | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character). See Remarks 15. and 20.   | Grid ID                               |
| PSDVELO               | Velocity Component (see Remarks 27. and 31.)        | Frequency Value<br>(Blank; Real $\geq 0.0$ or Character). See Remarks 15. and 20.    | Grid ID                               |
| PSDACCL               | Acceleration Component<br>(see Remarks 27. and 31.) | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character). See Remarks 15. and 20.   | Grid ID                               |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE)  | Response Attributes  |  |                            |
|--|--|--|----------------------------|
|  | ATTA (Integer > 0)   | ATTB (Integer > 0 or Real > 0.0)   | ATTI (Integer > 0)         |
| RMSDISP  | Displacement Component (see Remark 31.)  | RANDPS ID  | Grid ID                    |
| RMSVELO  | Velocity Component (see Remark 31.)  | RANDPS ID  | Grid ID                    |
| RMSACCL  | Acceleration Component (see Remark 31.)  | RANDPS ID  | Grid ID                    |
| ACPWR<br>(Acoustic Power radiated through a panel)<br>See Remark 34. | Blank  | Frequency value.<br>(Blank for all forcing frequency; Real $\geq 0.0$ )              | Blank                      |
| ACINTS<br>(Acoustic Intensity)                                       | Blank  | Frequency value.<br>(Blank for all forcing frequency; Real $\geq 0.0$ )              | Grid ID of wetted surface. |
| AFPRES<br>(Acoustic Pressure for AFPM)<br>See Remark 35.             | Acoustic Pressure Component (Integer = 1 or 7)   | Frequency value<br>(Blank for all forcing frequency; Real $\geq 0.0$ )               | Grid ID of AFPMID.         |
| AFINTS<br>(Acoustic Intensity for AFPM)<br>See Remark 35.            | Component Code -<br>0 - normal to AFPM,<br>1 - x-dir<br>2 - y-dir<br>3 - z-dir   | Frequency value.<br>(Blank for all forcing frequency; Real $\geq 0.0$ )              | Grid ID of AFPMID.         |
| AFVELO<br>Velocity for AFPM)<br>See Remark 35.                       | Component Code -<br>11 - Real/Mag in x-dir<br>12 - Real/Mag in y-dir<br>13 - Real/Mag in z-dir<br>71 - Img/Ph in x-dir<br>72 - Img/Ph in y-dir<br>73 - Img/Ph in z-dir | Frequency value.<br>(Blank for all forcing frequency; Real $\geq 0.0$ )              | Grid ID of AFPMID.         |
| AFPWR<br>(Acoustic Power for AFPM) See Remark 35.                    | Blank  | Frequency value.<br>(Blank for all forcing frequency; Real $\geq 0.0$ )              | Blank                      |
| ERP<br>See Remarks 41. and 42.                                       | ERP Item Code  | Frequency Value.<br>(Blank; Real $\geq 0.0$ or Character)<br>See Remarks 15. and 20. | Set3 ID or Blank           |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE)             | Response Attributes         |  |                                       |
|-----------------------------------|-----------------------------|--|---------------------------------------|
|                                   | ATTI (Integer > 0)          | ATTB (Integer > 0 or Real > 0.0)                                   | ATTI (Integer > 0)                    |
| TDISP                             | Displacement Component      | Time Value. (Blank; Real; or Character)<br>See Remarks 16. and 20. | Grid ID                               |
| TVELO                             | Velocity Component          | Time Value. (Blank; Real, or Character)<br>See Remarks 16. and 20. | Grid ID                               |
| TACCL                             | Acceleration Component      | Time Value. (Blank; Real, or Character)<br>See Remarks 16. and 20. | Grid ID                               |
| TSPCF                             | SPC Force Component         | Time Value. (Blank; Real, or Character)<br>See Remarks 16. and 20. | Grid ID                               |
| TSTRE                             | Stress Item Code            | Time Value. (Blank; Real, or Character)<br>See Remarks 16. and 20. | Property ID (PID) or Element ID (EID) |
| TFORC                             | Force Item Code             | Time Value. (Blank; Real, or Character)<br>See Remarks 16. and 20. | Property ID (PID) or Element ID (EID) |
| STMONP1<br>Structural<br>MONPNT1  | Component<br>See Remark 36. | Blank  | Blank                                 |
| STMOND1<br>Structural<br>MONDSP1  | Component<br>See Remark 36. | Blank  | Blank                                 |
| MONPNT3                           | Component<br>See Remark 36. | Blank  | Blank                                 |
| AEMONP1<br>Aerodynamic<br>MONPNT1 | Component<br>See Remark 36. | Blank  | Blank                                 |

Table 12 Design Sensitivity Response Attributes

| Response Type (RTYPE)             | Response Attributes                      |  |                    |
|-----------------------------------|--|--|--------------------|
|                                   | ATTA (Integer > 0)                       | ATTB (Integer > 0 or Real > 0.0)                   | ATTI (Integer > 0) |
| AEMOND1<br>Aerodynamic<br>MONDSP1 | Component<br>See Remark 36.              | Blank  | Blank              |
| TRIM                              | AESTAT or AESURF entry ID                | Blank  | Blank              |
| STABDER                           | AESTAT or AESURF entry ID                | Restraint Flag.<br>(Integer 0 or 1) See Remark 13. | Component          |
| FLUTTER                           | Blank                                    | Blank  | See Remark 14.     |
| DIVERG                            | Divergence Root Number<br>See remark 38. | Blank  | Mach No.           |
| WMPID<br>See Remarks 44. and 45.  | MID                                      | SEID   | PID                |

**Remarks:**

1. Stress, strain, force, fatigue and ERP item codes can be found in [Item Codes, 1045](#). For stress or strain item codes that have dual meanings, such as von Mises or maximum shear, the option specified in the Case Control Section will be used; i.e., STRESS(VONM) or STRESS(MAXS).
2. RTYPE = "CSTRESS", "CSTRAIN", "CFAILURE", and "CSTRAT" are used only with the PCOMP/PCOMPG entry. "CSTRESS" and "CSTRAIN" item codes are described under Table 1 (Element Stress/Strain Item Codes) in [Item Codes, 1045](#). "CFAILURE" and "CSTRAT" item codes are described under Table 2 (Element Force Item Codes) in [Item Codes, 1045](#). Only force item codes that refer to failure indices of direct stress and interlaminar shear stress are valid.

The CFAILURE and CSTRAT response types requires the following specifications on the applicable entries:

- Failure theory in the FT field on PCOMP/PCOMPG entry.
  - Allowable bonding shear stress in the SB field on PCOMP/PCOMPG entry.
  - Stress limits in the ST, SC, and SS fields on all MATi entries.
  - When the PCOMPG is invoked, there is no default for the ATTB field.
3. ATTB can be used for responses of weight, composite laminae, dynamics, real and complex eigenvalues, WMPID and stability derivatives. For eigenvector responses, such as DISP, the ATTB identifies the mode of interest. For other responses, this field must be blank.
  4. All grids associated with a DRESP1 entry are considered to be in the same region for screening purposes. Only up to NSTR displacement constraints (see DSCREEN entry) per group per load case will be retained in the design optimization phase.

5. DRESP1 identification numbers must be unique with respect to DRESP2 identification numbers.
6. If PTYPE = "ELEM", the ATT<sub>i</sub> correspond to element identification numbers.
7. If RTYPE = "DISP", "SPCFORCE", "GPFORCE", "TDISP", "TVELO", "TACCL" or "TSPCF", multiple component numbers (any unique combination of the digits 1 through 6 with no embedded blanks) may be specified on a single entry. Multiple response components may not be used on any other response types.
8. If RTYPE = "FRDISP", "FRVELO", "FRACCL", or "FRSPCF" only one component number may be specified in the ATT<sub>A</sub> field. Numbers 1 through 6 correspond to real (or magnitude) components and 7 through 12 imaginary (or phase) components. If more than one component for the same grid is desired, then a separate entry is required.
9. Real/imaginary representation is the default for complex response types. Magnitude/phase representation must be requested by the corresponding Case Control command; e.g., DISP(PHASE) = ALL for FRDISP type responses.
10. REGION is used for constraint screening. The NSTR field on DSCREEN entries gives the maximum number of constraints retained for each region per load case.

IF RTYPE = "WEIGHT", "VOLUME", "LAMA", "EIGN", "FREQ", "CEIG", "TOTSE", "RMSDISP", "RMSVELO", "RMSACCL", no REGION identification number should be specified. If the region field is left blank for a grid response, one region is created for each DRESP1 ID. If the region field is left blank for an element response, one region is created for each property ID invoked. Usually, the default value is appropriate.

If the REGION field is not blank, all the responses on this entry as well as all responses on other DRESP1 entries that have the same RTYPE and REGION identification number will be grouped into the same region.
11. REGION is valid only among the same type of responses. Responses of different types will never be grouped into the same region, even if they are assigned the same REGION identification number by the user.
12. If RTYPE = "WEIGHT", "VOLUME", or "TOTSE" field ATT<sub>i</sub> = "ALL" implies total weight/volume/total strain energy of all superelements except external superelements, 0 implies residual only and i implies SEID=i. Default="ALL". RTYPE="TOTSE" is not supported for shape optimization.
13. RTYPE = "STABDER" identifies a stability derivative response. ATT<sub>B</sub> is the restraint flag for the stability derivative. ATT<sub>B</sub> = 0 means unrestrained, and ATT<sub>B</sub> = 1 means restrained. For example, ATT<sub>A</sub> = 4000, ATT<sub>B</sub> = 0, and ATT<sub>1</sub> = 3 reference the unrestrained C<sub>z</sub> derivative for the AESTAT (or AESURF) entry ID = 4000.
14. RTYPE = "FLUTTER" identifies a set of damping responses. The set is specified by ATT<sub>i</sub>:

ATT<sub>1</sub> = Identification number of a SET1 entry that specifies a set of modes.

ATT<sub>2</sub> = Identification number of an FLFACT entry that specifies a list of densities.

ATT<sub>3</sub> = Identification number of an FLFACT entry that specifies a list of Mach numbers.

ATT<sub>4</sub> = Identification number of an FLFACT entry that specifies a list of velocities.

If the flutter analysis is type PKNL, it is necessary to put PKNL in the PTYPE field of this entry.

15. For RTYPE = "FRXXXX", "PSDXXXX", "PRES" and "ERP" a real value for ATTB specifies a frequency value in cycles per unit time. If a real ATTB value is specified, then the responses are evaluated at the closest excitation frequency. The default for ATTB is all excitation frequencies. See Remark 20. for additional ATTB options. The OFREQ Case Control command has no effect on the selection of the frequencies.
16. For RTYPE = "TDISP", "TVELO", "TACCL", "TSPCF", "TFORC", and "TSTRE", ATTB specifies a time value. If ATTB is specified, then the responses are evaluated at the closest time selected by the OTIME command. The default for ATTB is all time steps selected by the OTIME command.
17. Intermediate station responses on CBAR elements due to PLOAD1 and/or CBARAO entries may not be defined on the DRESP1 entry.
18. RTYPE = "EIGN" refers to normal modes response in terms of eigenvalue (radian/time)\*\*2 while RTYPE = "FREQ" refers to normal modes response in terms of natural frequency or units of cycles per unit time.
19. For RTYPE = LAMA, EIGN or FREQ, the response approximation used for optimization can be individually selected using the ATTB field when APRCOD = 1 is being used.

For RTYPE = LAMA, ATTB = blank or 1 selects direct linearization, ATTB = 2 = inverse linearization.

For RTYPE = EIGEN or FREQ, ATTB = blank = Rayleigh Quotient Approximation, = 1 = direct linearization, = 2 = inverse approximation.

The default Rayleigh Quotient Approximation should be preferred in most cases.

20. Character input for ATTB is available for RTYPE of "FRXXXX", "PSDXXXX", "TXXXX", "PRES" and "ERP". The character input represents a mathematical function and the options for character input are SUM, AVG, SSQ, RSS, MAX and MIN. The expression of mathematical function is shown as follows:

$$\text{SUM}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i$$

$$\text{AVG}(X_1, X_2, \dots, X_n) = \frac{\sum_{i=1}^n X_i}{n}$$

$$\text{SSQ}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i^2$$

$$\text{RSS}(X_1, X_2, \dots, X_n) = \sqrt{\sum_{i=1}^n X_i^2}$$

$$\text{MAX}(X_1, X_2, \dots, X_n) = \text{Maximum value among } X_i(i=1 \text{ to } n)$$

$$\text{MIN}(X_1, X_2, \dots, X_n) = \text{Minimum value among } X_i(i=1 \text{ to } n)$$

where  $X_i$  is the response for a forcing frequency or time step. For example

DRESP1,10,DX1,FRSTRE,ELEM,,3,AVG,10

yields a response which is equal to the average stress for element 10 across all forcing frequencies.

**CAUTION!**

1. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP2" flag, the results of the mathematical function are used to evaluate the DRESP2.
2. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP1" flag, the individual values of the DRESP1 are used to compute individual DRESP2 values and the mathematical function is not used by the DRESP2.

21. Element strain energy item codes can be found under [Table 7-13](#) in [Item Codes, 1045](#). Only element strain energy and element strain energy density can be referenced on a DRESP1 entry. RTYPE="ESE" is not supported for shape optimization.
22. For RTYPE=RMSDISP, RMSVELO, or RMSACCL the ATTB specifies the appropriate RANDPS ID.
23. Input other than 1 or 7 of ATTA field, acoustic pressure component, for PRES response type will be reset to 1 (if less than 7) or 7 (if greater than 6 and less than 13).
24. Design response weight is obtained from Grid Point Weight Generator for a reference point GRDPNT (see parameter GRDPNT). If GRDPNT is either not defined, equal to zero, or not a defined grid point, the reference point is taken as the origin of the basic coordinate system. Fields ATTA and ATTB refer to the row and column numbers of the rigid body weight matrix, which is partitioned as follows:

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6 \times 6}$$

The default values of ATTA and ATTB are 3, which specifies weight in the Z direction. Field ATT1 = "ALL" implies total weight of all superelements except external superelements. SEIDI refers to a superelement identification number. SEIDI = "0" refers to the residual superelement. The default of ATT1 is blank which is equivalent to "ALL".

25. For RTYPE = GPFORCE, the PTYPE field is used to designate the GRID ID at which the force is defined. Output that is produced using PARAM NOELOF > 0 is not supported for the DRESP1 entry.

26. For RTYPE = GPFORCP, the PTYPE field is blank. The grid point force is for the sum of all elements from the GRID ID listed in ATTA to the GRID (orient ID) listed in ATTi. This response corresponds to that produced with PARAM NOELP > 0. It is not necessary to set PARAM NOELP > 0 to compute the GPFORCP response.
27. For RTYPE = PSDXXXX, the PTYPE field specifies the RANDPS ID.
28. RTYPE=COMP (compliance of structures =  $P^T u$ ) and FRMASS (mass fraction of topology designed elements) entries are used for topology optimization or combined topology, sizing/shape optimization.
29. RTYPE=FRMASS can be used for topology and topometry optimization. For topology, FRMASS, is the mass divided by the mass calculated if all topology design variables are 1.0. FRMASS is calculated for designed elements only. FRMASS = 1.0 if all design variables are 1.0. For topometry, the initial FRMASS is defined as 1.0 for the initial design specified on the TOMVAR entries. For non-volume elements like CELAS, an artificial mass = 1.0 is assumed for each element.
30. Response type ABSTRESS is for element type code of 238 (CBAR) and 239 (CBEAM) only.
31. If a PSDxxxx or RMSxxxx response is being applied with RANDPS Bulk Data entries that include multiple subcases, the DESSUB or DESOBJ Case Control command that invokes the PSDxxxx or RMSxxxx response must be in the first subcase used by the RANDPS entry.
32. Response types, ESE, TOTSE, GPFORCE and GPFORCP are not supported for shape optimization.
33. For RTYPE=EIGN or FREQ, PTYPE field can be utilized to identify the source of the mode. Valid options are 'STRUC' or 'FLUID'. The default is 'STRUC'.
34. For RTYPE=ACPWR, PTYPE field can be utilized to identify the panel name. The default is 'total' which covers the whole interface between structural and fluid field.
35. For RTYPE=AFxxxx, PTYPE field can be utilized to specified the ID of Acoustic Field Point Mesh, AFPM.
36. For monitor point responses (RTYPE = STMONP1, STMOND1, MONPNT3 AEMONP1 or AEMOND1) the ATTA field specifies the components to be extracted. These can be any subset of the integers 1 through 6 that appear on the monitor quantity with the NAME provided in the PTYPE field. All of these responses can be invoked in a static aeroelastic (ANALYSIS=SAERO) subcase. STMOND1 and MONPNT3 can be invoked from a static (ANALYSIS=STAT) subcase. The responses are not available in a dynamic response or normal modes subcase. The response types have the following meaning:
  - a. STMONP1 – A structural MONPNT1
  - b. STMOND1 – A structural MONDSP1
  - c. MONPNT3 – A MONPNT3
  - d. AEMONP1 – An aerodynamic MONPNT1
  - e. AEMOND1 – An aerodynamic MONDSP1

For all but the STMONP1, the response is the elastic monitor point value. For the STMONP1, it is the elastic "minus" inertial "plus" elastic applied load value.

37. For RTYPE-FRMASS, if the PID in the ATTi field is not blank, the PTYPE field must specify the type of property the PID references.
38. RTYPE=DIVERG is for subcases with ANALYSIS=DIVERG (aeroelastic static divergence). Only a single Mach number can be specified for this RTYPE.
39. For RTYPE = FATIGUE (pseudo-static fatigue), PTYPE must be set to ELEM or PSOLID, PSHELL, PBAR, PBEAM, or PWELD. For RTYPE = FRFTG (frequency response fatigue), PTYPE must be set to ELEM or PSOLID or PSHELL. ATTB is the FID of a FATIGUE case control and must be the same FID for all RTYPES of type FATIGUE or FRFTG. RTYPES of type FATIGUE and FRFTG cannot be mixed; either all are RTYPE=FATIGUE or all are RTYPE=FRFTG.
40. If a FATIGUE case control references a SET ID, then ATTB must reference one of the IDs referenced by the SET. Design optimization using fatigue responses is currently limited to one fatigue analysis, that is, one loading sequence. Multiple fatigue responses referencing different loading sequences, i.e., FATIGUE case control IDs, will result in an error.
41. For the ERP response, the PTYPE field must be ERPPNL.
42. If the ATT1 field is blank for the ERP response, all panels are invoked.
43. Fatigue Item Codes can be found in [Fatigue Item Codes, 1104](#). Item codes that are available as design responses are 4-9 and 12 for the element centroid or first node/layer/location and the comparable items for ensuing nodes/layers/locations. The use of NODA on the FTGPARM entry is not supported.
44. For the WMPID (weight as a function of material and property ID), the following conventions apply:
  - a. ATTA is required, ATTB and ATTi are optional.
  - b. ATTB specifies the superelement with ATTB=0 indicating the residual, >0, indicates superelement ATTB and ATTB = ALL or blank means all superelements)
  - c. The ATTi fields designate the property ID's of interest. If ATTi is specified, the associated property type (e.g., PSHELL) must be specified by the PTYPE input of field 5 on the DRESP1 entry.
  - d. If there is no ATTi, all material with MID=ATTa will be used to generate the response.
  - e. If there are multiple ATTi, a single scalar weight is calculated that is the sum of the all the properties
  - f. Materials that are supported with this entry are: MAT1,MAT2, MAT3, MAT8, MAT9 and MAT10
  - g. Properties that are supported are: PSHELL, PROD, PBEAM, PTUBE, PSHEAR, PBAR, PSOLID, PBEND,PCOMP, PCOMPG, PBARL, PBEAML, PBRSECT and PBMSECT.
45. Certain restrictions apply for RTYPE=WMPID
  - a. The density of the material referenced on the ATTA field cannot also be designed using a DVMRELx entry
  - b. The property referenced on the ATTi field cannot also be invoked by a TOMVAR entry.
  - c. The WMPID response is supported for topology optimization or for shape sizing optimization, but not for both topology and shape sizing optimization.

**DRESP2****Design Sensitivity Equation Response Quantities**

Defines equation responses that are used in the design, either as constraints or as an objective.

**Format:**

| 1      | 2         | 3      | 4            | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|-----------|--------|--------------|--------|--------|--------|--------|--------|----|
| DRESP2 | ID        | LABEL  | EQID or FUNC | REGION | METHOD | C1     | C2     | C3     |    |
|        | "DESVAR"  | DVID1  | DVID2        | DVID3  | DVID4  | DVID5  | DVID6  | DVID7  |    |
|        |           | DVID8  | -etc.-       |        |        |        |        |        |    |
|        | "DTABLE"  | LABL1  | LABL2        | LABL3  | LABL4  | LABL5  | LABL6  | LABL7  |    |
|        |           | LABL8  | -etc.-       |        |        |        |        |        |    |
|        | "DRESP1"  | NR1    | NR2          | NR3    | NR4    | NR5    | NR6    | NR7    |    |
|        |           | NR8    | -etc.-       |        |        |        |        |        |    |
|        | "DNODE"   | G1     | C1           | G2     | C2     | G3     | C3     |        |    |
|        |           | G4     | C4           | etc.   |        |        |        |        |    |
|        | "DVPREL1" | DPIP1  | DPIP2        | DPIP3  | DPIP4  | DPIP5  | DPIP6  | DPIP7  |    |
|        |           | DPIP8  | DPIP9        | -etc.- |        |        |        |        |    |
|        | "DVCREL1" | DCIC1  | DCIC2        | DCIC3  | DCIC4  | DCIC5  | DCIC6  | DCIC7  |    |
|        |           | DCIC8  | DCIC9        | -etc.- |        |        |        |        |    |
|        | "DVMREL1" | DMIM1  | DMIM2        | DMIM3  | DMIM4  | DMIM5  | DMIM6  | DMIM7  |    |
|        |           | DMIM8  | DMIM9        | -etc.- |        |        |        |        |    |
|        | "DVPREL2" | DPI2P1 | DPI2P2       | DPI2P3 | DPI2P4 | DPI2P5 | DPI2P6 | DPI2P7 |    |
|        |           | DPI2P8 | DPI2P9       | -etc.- |        |        |        |        |    |
|        | "DVCREL2" | DCI2C1 | DCI2C2       | DCI2C3 | DCI2C4 | DCI2C5 | DCI2C6 | DCI2C7 |    |
|        |           | DCI2C8 | DCI2C9       | -etc.- |        |        |        |        |    |
|        | "DVMREL2" | DMI2M1 | DMI2M2       | DMI2M3 | DMI2M4 | DMI2M5 | DMI2M6 | DMI2M7 |    |
|        |           | DMI2M8 | DMI2M9       | -etc.- |        |        |        |        |    |
|        | "DRESP2"  | NRR1   | NRR2         | NRR3   | NRR4   | NRR5   | NRR6   | NRR7   |    |
|        |           | NRR8   | -etc.-       |        |        |        |        |        |    |
|        | "DVLREL1" | DLIL1  | DLIL2        | DLIL3  | DLIL4  | DLIL5  | DLIL6  | DLIL7  |    |
|        |           | DLIL8  | -etc.-       |        |        |        |        |        |    |

**Example:**

|        |        |       |    |   |    |   |     |     |  |
|--------|--------|-------|----|---|----|---|-----|-----|--|
| DRESP2 | 1      | LBUCK | 5  | 3 |    |   |     |     |  |
|        | DESVAR | 101   | 3  | 4 | 5  | 1 | 205 | 209 |  |
|        |        | 201   |    |   |    |   |     |     |  |
|        | DTABLE | PI    | YM | L |    |   |     |     |  |
|        | DRESP1 | 14    | 1  | 4 | 22 | 6 | 33  | 2   |  |

|  |         |     |     |     |   |    |   |  |  |
|--|---------|-----|-----|-----|---|----|---|--|--|
|  | DNODE   | 14  | 1   | 4   | 1 | 22 | 3 |  |  |
|  |         | 2   | 1   | 43  | 1 |    |   |  |  |
|  | DVPREL1 | 101 | 102 |     |   |    |   |  |  |
|  | DVCREL1 | 201 | 202 |     |   |    |   |  |  |
|  | DVMREL1 | 301 |     |     |   |    |   |  |  |
|  | DVPREL2 | 401 | 402 |     |   |    |   |  |  |
|  | DVCREL2 | 501 |     |     |   |    |   |  |  |
|  | DVMREL2 | 601 | 602 | 603 |   |    |   |  |  |
|  | DRESP2  | 50  | 51  |     |   |    |   |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number. (Integer > 0)   |
| LABEL      | User-defined label. (Character, no default)   |
| EQID       | DEQATN entry identification number. (Integer > 0)   |
| FUNC       | Function to be applied to the arguments. See Remark 8. (Character)  |
| REGION     | Region identifier for constraint screening. See Remark 5. (Integer > 0)   |
| METHOD     | When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN)   |
|            | When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS)                                |
| Ci         | Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. (Real; Defaults: C1 = 1.0., C2 = .005, and C3=10.0)                                    |
| “DESVAR”   | Flag indicating DESVAR entry identification numbers. (Character)  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)   |
| “DTABLE”   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character) |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)  |
| “DRESP1”   | Flag indicating DRESP1 entry identification numbers. (Character). See Remark 13.  |
| NRk        | DRESP1 entry identification number. (Integer > 0)   |
| “DNODE”    | Flag indicating grid point and component identification numbers. (Character)  |
| Gm         | Identification number for any grid point in the model. (Integer > 0)  |
| Cm         | Component number of grid point Gm. ( $1 \leq \text{Integer} \leq 3$ )   |
| “DVPREL1”  | Flag indicating DVPREL1 entry identification number. (Character)  |
| DPIPi      | DVPREL1 entry identification number. (Integer > 0)  |

| Descriptor | Meaning   |
|------------|---|
| “DVCREL1”  | Flag indicating DVCREL1 entry identification number. (Character)                      |
| DCICi      | DVCREL1 entry identification number. (Integer > 0)                                    |
| “DVMREL1”  | Flag indicating DVREL1 entry identification number. (Character)                       |
| DMIMi      | DVMREL1 entry identification number. (Integer > 0)                                    |
| “DVPREL2”  | Flag indicating DVPREL2 entry identification number. (Character)                      |
| DPI2Pi     | DVPREL2 entry identification number. (Integer > 0)                                    |
| “DVCREL2”  | Flag indicating DVCREL2 entry identification number. (Character)                      |
| DCI2Ci     | DVCREL2 entry identification number. (Integer > 0)                                    |
| “DVMREL2”  | Flag indicating DVMREL2 entry identification number. (Character)                      |
| DMI2Mi     | DVMREL2 entry identification number. (Integer > 0)                                    |
| “DRESP2”   | Flag indicating other DRESP2 entry identification number. (Character). See Remark 13. |
| NRRk       | DRESP2 entry identification number. (Integer > 0)                                     |
| “DVLREL”   | Flag indicating DVLREL1 identification numbers. (Character)                           |
| DLILi      | DVLREL1 entry identification number. (Integer > 0)                                    |

#### Remarks:

1. DRESP2 entries may only reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2 and DVLREL1 entries. They may also reference other DRESP2 entries. However, a DRESP2 entry cannot reference itself directly or recursively.
2. a) If the referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify DRESP1 IDs for each subcase. DRESP2 entries that span subcases must be invoked above the subcase level by DESGLB on DESOBJ commands.  
b) Referenced DRESP entries that span superelements are supported automatically.  
c) Referenced DRESP2 entries cannot span subcases or superelements.
3. DRESP2 entries must have unique identification numbers with respect to DRESP1 entries.
4. The “DESVAR”, “DTABLE”, “DRESP1”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2” and DVLREL1 flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP2 relationship. However, at least one of these ten types of arguments must exist.
5. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP2 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP2 entry in the same region.

6. The variables identified by DVIDi, LABLj, NRk, the Gm, CMPM pairs, DPPIP, DCICm, DMIMn, DPI2Po, DCI2Cp, DMI2Mq, and NRRu are assigned (in that order) to the variable names (x1, x2, x3, etc.) specified in the left-hand side of the first equation on the DEQATN entry referenced by EQID. In the example below,

DESVARs 101 and 3 are assigned to arguments A and B.  
 DTABLEs PI and YM are assigned to arguments C and D.  
 Grid 14, Component 1 is assigned to argument R.

|        |        |  |    |   |  |  |  |  |  |
|--------|--------|--|----|---|--|--|--|--|--|
| DRESP2 | 1      | LBUCK  | 5  | 3 |  |  |  |  |  |
|        | DESVAR | 101  | 3  |   |  |  |  |  |  |
|        | DTABLE | PI   | YM |   |  |  |  |  |  |
|        | DNODE  | 14   | 1  |   |  |  |  |  |  |
| DEQATN | 5      | $F1(A, B, C, D, R)=A+B*C-(D^{**}3+10.0)+\sin(C*R)$ |    |   |  |  |  |  |  |

7. (Gm, Cm) can refer to any grid component and is no longer limited to a designed grid component.  
 8. The FUNC attributes can be used in place of the EQID and supports the functions shown in the following table:

| Function | Description   |
|----------|---|
| SUM      | Sum of the arguments  |
| SFMAX    | Maximization of stiffness and frequency. See remark 14.           |
| AVG      | Average of the arguments  |
| SSQ      | Sum of the squares of the arguments                               |
| RSS      | Square root of the sum of the squares of the arguments            |
| MAX      | The maximum value of the argument list                            |
| MIN      | The minimum value of the argument list                            |
| BETA     | Minimize the maximum response. See Remark 10.                     |
| MATCH    | Match analysis results with user specified values. See Remark 11. |

When EQID has character input, the DEQATN entry is no longer needed. The functions are applied to all arguments on the DRESP2 regardless of the type. See Remark 20. of the DRESP1 entry for an explanation of these functions.

9. The number of arguments of a DEQATN can be more than the number of values defined on the DRESP2 if the DRESP1s referenced have RTYPE with 'FR' or 'PSD' prefix. Arguments are still positional. The extra arguments in the DEQATN must appear at the end of the argument list. The discrepancy is resolved internally with the forcing frequency(ies) associated with DRESP1s. An example is shown as follows:

|        |    |        |        |  |  |   |     |      |  |
|--------|----|--------|--------|--|--|---|-----|------|--|
| DRESP1 | 10 | FDISP1 | FRDISP |  |  | 1 | 10. | 1001 |  |
| DRESP1 | 20 | FDISP2 | FRDISP |  |  | 1 | 20. | 1001 |  |

|        |        |                                      |     |  |  |  |  |  |
|--------|--------|--------------------------------------|-----|--|--|--|--|--|
| DRESP2 | 30     | AVGFD                                | 100 |  |  |  |  |  |
|        | DRESP1 | 10                                   | 20  |  |  |  |  |  |
| DEQATN | 100    | AVG(D1,D2,F1,F2) = (D1/F1+D2/F2)*0.5 |     |  |  |  |  |  |

In the above example, the DEQATN has two more additional terms than have been defined on the DRESP2. The first additional term is the forcing frequency (in hertz) of the first DRESP1 ID on the DRESP2. The second additional term is the forcing frequency of second DRESP1 ID in the list. When all DRESP1s involved have the same frequency, the user is not required to name all the additional terms in the argument list of DEQATN.

10. FUNC = BETA facilitates a design task where the objective is to minimize the maximum response. Only DRESP1 entries can be invoked by DRESP2 and the DRESP's cannot span subcases. The BETA function creates the following design task:

$$\text{Minimize } \phi = C_1 X_\beta$$

$$\text{Subject to } g = \frac{\gamma X_\beta - r_j}{C_3} \geq 0$$

where  $\gamma$  is determined from

$$C_2 = (r_{j\max} - \gamma X_\beta) / C_3$$

User input parameters  $C_1$ ,  $C_2$ ,  $C_3$  therefore have the following meaning:

$C_1$  (Default = 1.0) weights the spawned design variable,  $X_\beta$ , to create the objective. Since  $X_\beta$  starts at 1.0,  $C_1$  is the initial objective.

$C_2$  sets the initial value of the maximum constraint created by this process. The default values of 0.005 is equal to DOPTPRM parameter GMAX.

$C_3$  (Default = 10.0) is an offset value to avoid dividing by zero when creating constraints.

11. FUNC = MATCH creates a response from the difference between analysis results,  $r_j$ , that are associated with DRESP1s and target values,  $r_j^T$ , that are input using DTABLE data. Only DRESP1 entries and DTABLE entries can be invoked by the DRESP2 entry.

When METHOD = LS, a least square minimization is performed where the response is

$$r_2 = \sum_{j=1}^m \left( \frac{r_j - r_j^T}{|r_j^T|} \right)^2$$

When METHOD = BETA, the design task becomes one of minimizing an objective that is the maximum normalized difference between the analysis and target values

$$\frac{r_j - r_j^T}{|r_j^T|}$$

in the same manner as outlined in Remark 10.

12. With FUNC=MATCH, if the DTABLE LABLj invokes an integer VALUi on the DTABLE entry, the integer points to a TABLEDi entry that provides tabular input as a function of frequency or time. This is used in conjunction with a single DRESP1 NRk that provides a response across a range of frequencies or times (i.e., RTYPE=FRxxxx, PSDxxxx, ACxxxx or Txxxx). When the TABLEDi option is invoked, the response can only be constrained, it cannot be the objective. This further implies that FUNC=MATCH, METHOD=BETA is not supported with TABLEDi.
13. If the mathematical function from a DRESP1 that has character input in the ATTB field (see remark 20 of the DRESP1) is to be used in the DRESP2 evaluation, the DRESP1 must be referenced under the “DRESP2” flag, not the “DRESP1” flag. If the “DRESP1” flag is used in this situation, it will result in a DRESP2 being evaluated for each of the individual values that contribute to the mathematical function.
14. The functions are applied to all DRESP1 arguments on the DRESP2: Maximization of stiffness and frequency FUNC=SFMAX, that refers multiple DRESP1 IDs where the fundamental frequency ID must be placed at last one. In addition, the ANALYSIS = MODES must be placed at the last subcase in Case Control Section. This is typically used in topology, topometry and topography optimization.

**DRESP3**

## Design Sensitivity Response using user-supplied routine(s)

Defines an external response using user-supplied routine(s).

**Format:**

| 1      | 2         | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|-----------|--------|--------|--------|--------|--------|--------|--------|----|
| DRESP3 | ID        | LABEL  | GROUP  | TYPE   | REGION |        |        |        |    |
|        | "DESVAR"  | DVID1  | DVID2  | DVID3  | DVID4  | DVID5  | DVID6  | DVID7  |    |
|        |           | DVID8  | etc.   |        |        |        |        |        |    |
|        | "DTABLE"  | LABL1  | LABL2  | LABL3  | LABL4  | LABL5  | LABL6  | LABL7  |    |
|        |           | LABL8  | etc.   |        |        |        |        |        |    |
|        | "DRESP1"  | NR1    | NR2    | NR3    | NR4    | NR5    | NR6    | NR7    |    |
|        |           | NR8    | etc.   |        |        |        |        |        |    |
|        | "DNODE"   | G1     | C1     | G2     | C2     | G3     | C3     |        |    |
|        |           | G4     | C4     | etc.   |        |        |        |        |    |
|        | "DVPREL1" | DPIP1  | DPIP2  | DPIP3  | DPIP4  | DPIP5  | DPIP6  | DPIP7  |    |
|        |           | DPIP8  | DPIP9  | etc.   |        |        |        |        |    |
|        | "DVCREL1" | DCIC1  | DCIC2  | DCIC3  | DCIC4  | DCIC5  | DCIC6  | DCIC7  |    |
|        |           | DCIC8  | DCIC9  | -etc.- |        |        |        |        |    |
|        | "DVMREL1" | DMIM1  | DMIM2  | DMIM3  | DMIM4  | DMIM5  | DMIM6  | DMIM7  |    |
|        |           | DMIM8  | DMIM9  | -etc.- |        |        |        |        |    |
|        | "DVPREL2" | DPI2P1 | DPI2P2 | DPI2P3 | DPI2P4 | DPI2P5 | DPI2P6 | DPI2P7 |    |
|        |           | DPI2P8 | DPI2P9 | -etc.- |        |        |        |        |    |
|        | 'DCREL2"  | DCI2C1 | DCI2C2 | DCI2C3 | DCI2C4 | DCI2C5 | DCI2C6 | DCI2C7 |    |
|        |           | DCI2C8 | DCI2C9 | -etc.- |        |        |        |        |    |
|        | "DVMREL2" | DMI2M1 | DMI2M2 | DMI2M3 | DMI2M4 | DMI2M5 | DMI2M6 | DMI2M7 |    |
|        |           | DMI2M8 | DMI2M9 | -etc.- |        |        |        |        |    |
|        | "DRESP2"  | NRR1   | NRR2   | NRR3   | NRR4   | NRR5   | NRR6   | NRR7   |    |
|        |           | NRR8   | -etc.- |        |        |        |        |        |    |
|        | DVLREL1   | DLIL1  | DLIL2  | DLIL3  | DLIL4  | DLIL5  | DLIL6  | DLIL7  |    |
|        |           | DLIL8  | DLIL9  | -etc.- |        |        |        |        |    |
|        | "USRDATA" | String |        |        |        |        |        |        |    |
|        |           | -etc.- |        |        |        |        |        |        |    |

**Example:**

|        |        |       |         |      |   |   |     |     |  |
|--------|--------|-------|---------|------|---|---|-----|-----|--|
| DRESP3 | 1      | LBUCK | TAILWNG | BUCK |   |   |     |     |  |
|        | DESVAR | 101   | 3       | 4    | 5 | 1 | 205 | 209 |  |

|  |         |                              |     |     |    |    |    |   |  |
|--|---------|------------------------------|-----|-----|----|----|----|---|--|
|  |         | 201                          |     |     |    |    |    |   |  |
|  | DTABLE  | PI                           | YM  | L   |    |    |    |   |  |
|  | DRESP1  | 14                           | 1   | 4   | 22 | 6  | 33 | 2 |  |
|  | DNODE   | 14                           | 1   | 4   | 1  | 22 | 3  |   |  |
|  |         | 2                            | 1   | 43  | 1  |    |    |   |  |
|  | DVPREL1 | 101                          | 102 |     |    |    |    |   |  |
|  | DVCREL1 | 201                          | 202 |     |    |    |    |   |  |
|  | DVMREL1 | 301                          |     |     |    |    |    |   |  |
|  | DVPREL2 | 401                          | 402 |     |    |    |    |   |  |
|  | DVCREL2 | 501                          |     |     |    |    |    |   |  |
|  | DVMREL2 | 601                          | 602 | 603 |    |    |    |   |  |
|  | DRESP2  | 50                           | 51  |     |    |    |    |   |  |
|  | USRDATA | Constants: 12345.6 789.0 99. |     |     |    |    |    |   |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number. (Integer > 0)   |
| LABEL      | User-defined label. (Character, no default)   |
| GROUP      | Group name the external response type belongs to (Character). See Remark 2.   |
| TYPE       | External response type (Character). See Remark 3.   |
| “DESVAR”   | Flag indicating DESVAR entry identification numbers. (Character)  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)   |
| “DTABLE”   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character) |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)  |
| “DRESP1”   | Flag indicating DRESP1 entry identification numbers. (Character)  |
| NRk        | DRESP1 entry identification number. (Integer > 0)   |
| “DNODE”    | Flag signifying that the following fields are grid points.  |
| Gm         | Grid point identification number. (Integer > 0)   |
| Cm         | Degree-of-freedom number of grid point Gm. ( $1 \leq \text{Integer} \leq 3$ )   |
| “DVPREL1”  | Flag indicating DVPREL1 entry identification number. (Character)  |
| DPIPi      | DVPREL1 entry identification number. (Integer > 0)  |
| “DVCREL1”  | Flag indicating DVCREL1 entry identification number. (Character)  |
| DCICi      | DVCREL1 entry identification number. (Integer > 0)  |
| “DVMREL1”  | Flag indicating DVMREL1 entry identification number. (Character)  |
| DMIMi      | DVMREL1 entry identification number. (Integer > 0)  |

| Descriptor | Meaning   |
|------------|---|
| “DVPREL2”  | Flag indicating DVPREL2 entry identification number. (Character)      |
| DPI2Pi     | DVPREL2 entry identification number. (Integer > 0)                    |
| “DVCREL2”  | Flag indicating DVCREL2 entry identification number. (Character)      |
| DCI2Ci     | DVCREL2 entry identification number. (Integer > 0)                    |
| “DVMREL2”  | Flag indicating DVMREL2 entry identification number. (Character)      |
| DMI2Mi     | DVMREL2 entry identification number (Integer > 0)                     |
| “DRESP2”   | Flag indicating other DRESP2 entry identification number. (Character) |
| NRRk       | DRESP2 entry identification number. (Integer > 0)                     |
| “DVLREL1”  | Flag indicating DVLREL1 identification number (character)             |
| DLILi      | DVLREL1 entry identification number (integer)                         |
| “USRDATA”  | Flag indicating user input data (Character). See Remark 8.            |

#### Remarks:

1. DRESP3 entries may reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2, DRESP2 and DVLREL1 entries. However, a DRESP3 entry cannot reference another DRESP3 entry.
2. The group name must be referenced by an FMS CONNECT entry.
3. Multiple types of external responses can be defined in one group. Each type name identifies a specific external response evaluated in the user-supplied routines. See [Building and Using the Sample Programs MSC Nastran Utilities Guide](#) for a discussion of how to incorporate external responses.
4. a) Referenced DRESP2 entries cannot span subcases or superelements.  
b) If referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify the DRESP1 IDs for each subcase. DRESP3 entries that span subcases must be invoked above the subcase level by DESGLB or DESOBJ commands.  
c) Referenced DRESP1 entries that span superelements are supported automatically.
5. DRESP3 entries must have unique identification numbers with respect to DRESP2 and DRESP1 entries.
6. The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2”, “DVLREL1” and “USRDATA” keywords on the continuation entries must appear in the order given above. Any of these words, along with the subsequent data associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.
7. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.

8. The data in the USRDATA field is character string based. It provides a convenient way to pass constants to the external response server routines. The maximum number of characters allowed is 32000.

**DSCREEN****Design Constraint Screening Data**

Defines screening data for constraint deletion.

**Format:**

| 1       | 2     | 3   | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-------|-----|------|---|---|---|---|---|----|
| DSCREEN | RTYPE | TRS | NSTR |   |   |   |   |   |    |

**Example:**

|         |        |      |   |  |  |  |  |  |  |
|---------|--------|------|---|--|--|--|--|--|--|
| DSCREEN | STRESS | -0.7 | 2 |  |  |  |  |  |  |
|---------|--------|------|---|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| RTYPE      | Response type for which the screening criteria apply. See Remark 3. (Character)                                  |
| TRS        | Truncation threshold. (Real; Default = -0.5)   |
| NSTR       | Maximum number of constraints to be retained per region per load case. See Remark 3. (Integer > 0; Default = 20) |

**Remarks:**

1. Grid responses associated with one particular load case are grouped by the specification of DRESP1 entries. From each group, a maximum of NSTR constraints are retained per load case.
2. Element responses are grouped by the property; i.e., all element responses for one particular load case belonging to the set of PIDs specified under ATTi on a DRESPi entry are regarded as belonging to the same region. In superelement sensitivity analysis, if the property (PID) is defined in more than one superelement, then separate regions are defined. A particular stress constraint specification may be applied to many elements in a region generating many stress constraints, but only up to NSTR constraints per load case will be retained.
3. For aeroelastic responses, that is RTYPE = "TRIM", "STABDER", and "FLUTTER", the NSTR limit is applied to all DRESP1 IDs that are the same RTYPE and have the same REGION specified.
4. For responses that are not related to grids or elements, that is RTYPE = "WEIGHT", "VOLUME", "EIGN", "FREQ", "LAMA", CEIG", FRMASS, COMP, and TOTSE", NSTR is not used. TRS is still applicable.
5. The RTYPE field is set to EQUA if constraints that are associated with DRESP2 entries are to be screened. The RTYPE field is set to DRESP3 if constraints that are associated with DRESP3 entries are to be screened. If the REGION field on the DRESP2 or DRESP3 is blank, one region is established for each DRESP2/DRESP3 entry.
6. If a certain type of constraint exists but no corresponding DSCREEN entry is specified, all the screening criteria used for this type of constraint will be furnished by the default values.
7. Constraints can be retained only if they are greater than TRS. See the Remarks under the [DCONSTR](#), [1706](#) entry for a definition of constraint value.
8. Constraint screening is applied to each superelement.

**DTABLE****Table Constants**

Defines a table of real constants that are used in equations (see DEQATN entry).

**Format:**

| 1      | 2      | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|--------|--------|-------|-------|-------|-------|-------|-------|-------|----|
| DTABLE | LABL1  | VALU1 | LABL2 | VALU2 | LABL3 | VALU3 | LABL4 | VALU4 |    |
|        | LABL5  | VALU5 | LABL6 | VALU6 | LABL7 | VALU7 | LABL8 | VALU8 |    |
|        | -etc.- |       |       |       |       |       |       |       |    |

**Example:**

|        |    |       |   |      |   |       |  |  |  |
|--------|----|-------|---|------|---|-------|--|--|--|
| DTABLE | PI | 3.142 | H | 10.1 | E | 1.0E6 |  |  |  |
|        | G  | 5.5E5 | B | 100. |   |       |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| LABLi      | Label for the constant. (Character)                                  |
| VALUi      | Value of the constant or TABLEDx ID. (Real or Integer, respectively) |

**Remarks:**

1. Multiple DTABLE entries may be specified in the Bulk Data Section.
2. LABLi are referenced by the LABj on the DRESP2, DRESP3, DVCREL2, DVMREL2, or DVPREL2 entries.
3. Trailing blank fields are permitted at the end of each line of LABLi/VALUi pairs, but intermediate blanks are not. (See the example above for permitted trailing blanks.)
4. For PART SE, if LABLi is referenced on SEDRSP2 and/or SEDRSP3, DTABLE entries must be placed in a PART SE where companion design model entries, such as DESVAR, DRESP1 and etc, are available.
5. If the VALUi is an integer, this points to the ID of a TABLEDx Bulk Data entry that lists the constants as a function of frequency or time. See Remark 12. on the DRESP2 entry.
6. LABLi must be unique across all DTABLE and DTABLE2 entries.

**DTABLE2****Table Constants**

Defines real constants from a field of property, material or connections bulk data entries which then can be invoked by a DVxREL2, DRESP2, or DRESP3 entry.

**Format:**

| 1       | 2     | 3      | 4    | 5      | 6     | 7      | 8    | 9      | 10 |
|---------|-------|--------|------|--------|-------|--------|------|--------|----|
| DTABLE2 | LABL1 | PNAME1 | PID1 | FNAME1 | LABL2 | PNAME2 | PID2 | FNAME2 |    |
|         | LABL3 | PNAME3 | PID3 | FNAME3 |       |        |      |        |    |

**Example:**

|         |        |         |      |    |       |     |    |   |  |
|---------|--------|---------|------|----|-------|-----|----|---|--|
| DTABLE2 | PTHK10 | PSHELLL | 10   | T  | MATIE | MT1 | 38 | E |  |
|         | CBARX1 | CBAR    | 3888 | X1 |       |     |    |   |  |

| Descriptor | Meaning  |
|------------|--|
| LABLi      | Label for the constant. (Character)                                |
| PNAMEi     | Property, material or connection bulk data entry name. (Character) |
| PIDI       | ID of PNAMEi entry. (Integer > 0)                                  |
| FNAMEi     | Field name of PNAMEi. (Character)                                  |

**Remarks:**

1. LABLi on DTABLE2 and DTABLE must be unique.
2. LABLi on DTABLE2 can be referenced under DTABLE flag of DVxREL2 (where x=P, M or C)/DRESP2/DRESP3.
3. Values for the FNAMEi field of the PNAMEi Bulk Data entry with the ID of PIDI are taken from analysis model before updating of analysis values with the designed value. If the updated value is desired, use the DVxREL2 flag on DRESP2 or DRESP3 entries instead.
4. FNAMEi must be the same as the character string that appears on the PNAMEi Bulk Data entry.
5. DTABLE2 must be utilized along with 'NASTRAN SYSTEM(444)=1' in input file or 'sys444=1' during job submittal.

**DTI****Direct Table Input**

Defines table data blocks.

**Format:**

| 1   | 2    | 3    | 4      | 5   | 6      | 7        | 8  | 9  | 10 |
|-----|------|------|--------|-----|--------|----------|----|----|----|
| DTI | NAME | "0"  | T1     | T2  | T3     | T4       | T5 | T6 |    |
|     | V01  | V02  | -etc.- |     |        |          |    |    |    |
| DTI | NAME | IREC | V1     | V2  | V3     | V4       | V5 | V6 |    |
|     | V7   | V8   | V9     | V10 | -etc.- | "ENDREC" |    |    |    |

**Example:** (The first logical entry is the header entry.)

|     |     |      |     |    |      |       |        |   |  |
|-----|-----|------|-----|----|------|-------|--------|---|--|
| DTI | XXX | 0    | 3   | 4  | 4096 |       | 1      | 0 |  |
|     | 1.2 | 2.3  |     |    |      |       |        |   |  |
| DTI | XXX | 1    | 2.0 | -6 | ABC  | 6.000 | -1     | 2 |  |
|     | 4   | -6.2 | 2.9 | 1  | DEF  | -1    | ENDREC |   |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Any character string that will be used in the DMAP sequence to reference the data block. See Remark 1. (Character; the first character must be alphabetic.) |
| Ti         | Trailer values. (Integer $\geq 0$ ; Default = 32767)  |
| IREC       | Record number. (Integer $> 1$ )   |
| V0i, Vi    | Value. (Integer, Real, Character or blank)  |
| "ENDREC"   | Flags the end of the string of values (V0i or Vi) that constitute record IREC. (Character)  |

**Remarks:**

1. The user defines the data block and therefore must write a DMAP (or ALTER a solution sequence), which includes the DTIN modules, in order to use the DTI feature. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. All fields following ENDREC must be blank.
3. The entry using IREC = 0 is called the header entry and is an optional entry. The values T1 through T6 go to a special record called the trailer. Other values on the optional continuation go to the header record. If the header entry or the trailer is not specified, T1 through T6 = 32767. On this entry, "ENDREC" may be used only if there is at least one continuation.
4. In addition to the optional header entry, there must be one logical entry for each record in the table. Null records require no entries.

5. “ENDREC” is used to input blank values at the end of a record. If “ENDREC” is not specified, the string for a record ends with the last nonblank field.
6. The maximum number of DMI and DTI data blocks is 1000.
7. If Ti is not an integer, a machine-dependent error will be issued that may be difficult to interpret.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

**DTI,ESTDATA****Superelement Estimation Data Overrides**

Provides override data for time and space estimation for superelement processing operations.

**Format:**

| 1   | 2       | 3   | 4   | 5   | 6      | 7 | 8 | 9 | 10 |
|-----|---------|-----|-----|-----|--------|---|---|---|----|
| DTI | ESTDATA | "0" |     |     |        |   |   |   |    |
|     | kd1     | vd1 | kd2 | vd2 | -etc.- |   |   |   |    |

The next entries are repeated for any superelement for which estimate data overrides are desired. IREC must be incremented by 1.

|     |         |      |        |      |    |    |    |    |  |
|-----|---------|------|--------|------|----|----|----|----|--|
| DTI | ESTDATA | IREC | SEFLAG | SEID | k1 | v1 | k2 | v2 |  |
|     | k3      | v3   | -etc.- |      |    |    |    |    |  |

**Example:**

|     |         |     |    |    |    |     |    |     |  |
|-----|---------|-----|----|----|----|-----|----|-----|--|
| DTI | ESTDATA | 0   |    |    |    |     |    |     |  |
|     | NOMASS  | -1  |    |    |    |     |    |     |  |
| DTI | ESTDATA | 1   | SE | 10 | C1 | 5.5 | C3 | 4.5 |  |
|     | C7      | 7.3 |    |    |    |     |    |     |  |

| Descriptor | Meaning   |
|------------|---|
| kdi        | Keyword for estimation parameter. (Character from <a href="#">Table 9-9</a> .)  |
| vdi        | Value assigned to the estimation parameter kdi. (The type given in <a href="#">Table 9-9</a> .)                       |
| IREC       | Record number beginning with 1. (Integer > 0)   |
| SEFLAG     | SEFLAG = "SE" or "SEID" indicates the next field containing a superelement identification number. (Character)         |
| SEID       | Superelement identification number. (Integer > 0)   |
| ki         | Keyword for override of estimation parameter for indicated superelement. (Character from <a href="#">Table 9-9</a> .) |
| vi         | Value for keyword ki. (Type depends on ki as shown in the <a href="#">Table 9-9</a> .)                                |

Table 9-9 DTI,ESTDATA Input Parameter Descriptions

| Input Parameters |           |                   |             | Meaning and Comments  |
|------------------|-----------|-------------------|-------------|---|
| Keyword          | Type      | Default Value     | Math Symbol |   |
| CRMS*            | Real      | -1.0              | C           | Number of active columns in $[K_{oo}]$ .  |
| FCRMS*           | Real      | 0.10              |             | If FCRMS $\leq 0.0$ , FCRMS is used (c/o).  |
| C1               | Real      | 6.0               | $c_1$       | Average number of degrees-of-freedom per grid point in o-set.   |
| C3               | Real      | 8.0               | $c_3$       | Average number of connections per grid point.   |
| C4               | Real      | 0.15              | $c_4$       | I/O time (seconds) per block transferred.   |
| C5               | Real      | 6.0               | $c_5$       | Average number of effective degrees-of-freedom per grid point in a-set.   |
| C6               | Real      | 1.2               | $c_6$       | Total CPU factor.   |
| C7               | Real      | 6.0               | $c_7$       | Number of equivalent KGG data blocks for space estimation.  |
| WF               | Real      | -1.0              | W           | If WF $\leq 0.0$ then use available working storage in units of single-precision words.   |
| NOMASS           | Integer   | 1                 |             | If NOMASS $\neq 1$ then exclude mass terms from estimates.  |
| TSEX             | Real      | 0.5 (min)         |             | Threshold limit for CPU.  |
| SSEX             | Real      | 50.0 (blocks)     |             | Threshold limit for space.  |
| TWALLX           | Real      | 5.0 (min)         |             | Threshold limit for wall time.  |
| BUFSIZ           | Integer   | Machine Buffsize  | B           | Buffsize. See <a href="#">The NASTRAN Statement (Optional)</a> (p. 14) in the <i>MSC Nastran Reference Guide</i> .                                  |
| ML               | Real      | Machine Loop Time | M           | Arithmetic time for the multiply/add loop. See the <i>SOL 700 Explicit Nonlinear User's Guide</i> .   |
| CONIO            | Integer   | Machine I/O ratio |             | I/O count/CPU equivalence   |
| PREC             | Integer   | 1 or 2            |             | Machine Word Length (1 = long, 2 = short). See <a href="#">The NASTRAN Statement (Optional)</a> (p. 14) in the <i>MSC Nastran Reference Guide</i> . |
| NLOADS           | Integer   | 1                 | $N_L$       | Number of loading conditions  |
| SETYPE           | Character | "T"               |             | Superelement type (T = Tip)   |
| CMAX             | Real      | -1.0              | $C_{\max}$  | Maximum bandwidth   |

## Parameters Obtained from SEMAP

|     |                                 |
|-----|---------------------------------|
| NGI | Number of interior grid points. |
| NPE | Number of exterior grid points. |
| NS  | Number of scalar points         |
| NE  | Number of elements.             |

## Derived Parameters

|                         |                                     |
|-------------------------|-------------------------------------|
| $O = C1 + NGI$          | Size of o-set.                      |
| $A = C5(NPE - NS) + NS$ | Size of a-set.                      |
| $T = BUFSIZE/PREC$      | Number of matrix terms in a buffer. |

## Estimation Equations

For each superelement, estimates of CPU time and disk space are made using the following equations.

Table 9-10 Equations Used for CPU Time and Disk Space Estimate

| Printout Symbol | Math Symbol | Equations  |
|-----------------|-------------|--|
| TD              | $T_1$       | $T_1 = 1/2 \cdot M \cdot O \cdot C^2$  |
| TFBS            | $T_2$       | $T_2 = 2 \cdot M \cdot C \cdot O \cdot a$  |
| TMAA            | $T_3$       | $T_3 = M \cdot O \cdot a^2$ (set to 0.0 if NOMASS ≠ +1)  |
| TSE             | $T_{SE}$    | $T_{SE} = C_6(T_1 + T_2 + T_3)$  |
| SLOO            | $S_1$       | $S_1 = O \cdot C \cdot \frac{PREC}{WF}$  |
| SGO             | $S_2$       | $S_2 = O \cdot a \cdot \frac{PREC}{B}$   |
| SKGG            | $S_3$       | $S_3 = 36(NG_i + NG_e - NS)(C_3 + 1.0)\left(\frac{PREC}{B}\right)$                                       |
| SSE             | $S_{SE}$    | $S_{SE} = S_1 + S_2 + C_7 \cdot S_3$   |
| PASSES          | $p$         | FBS passes = $p = a \cdot O \cdot \frac{PREC}{WF}$   |
| BKSTRN          | $BT$        | Blocks Transferred = $BT = 2 \cdot p \cdot S_1 + S_2 + p \cdot S_2$ . (Last term omitted if NOMASS ≠ +1) |
| TWALL           | $T_W$       | Wall Time = $T_W = T_{SE} + C_4 \cdot BT$  |

## Remarks:

1. In the superelement solution sequences, this data is stored automatically.
2. The header record continuation entries are optional if no global override data is to be specified. In this case, the complete header entry is optional.
  - Active column data can come from one of several places. The value for CRMS is determined as follows:

- RMS from the entry when IREC > 0 and field 4 is “SE”.
  - RMS from entries with IREC = 0.
  - Computed bandwidth when PARAM,OLDSEQ is specified.
  - If FCRMS is specified when IREC > 0 and field 4 is “SE”, then CRMS = FCRMS · O .
  - If FCRMS is specified when IREC = 0, then CRMS = FCRMS · O .
  - CRMS = 0.1 · O .
3. If CMAX is not specified, then it is defaulted to CRMS.
  4. In the example above, mass terms are excluded for all superelements and new values are given for parameters C1, C3, and C7 for Superelement 10 only.
  5. The estimates for TSEX, SSEX, and TWALLX are not printed unless at least one estimate exceeds the threshold.

**DTI,INDTA****Stress, Strain and/or Force Sort/Filter Item Code Override**

Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.

**Format:**

| 1   | 2     | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-----|-------|-----|---|---|---|---|---|---|----|
| DTI | INDTA | "0" |   |   |   |   |   |   |    |

To specify/override items for a sort of stress quantities:

|     |       |     |    |    |    |    |          |  |  |
|-----|-------|-----|----|----|----|----|----------|--|--|
| DTI | INDTA | "1" | B1 | C1 | B2 | C2 | "ENDREC" |  |  |
|-----|-------|-----|----|----|----|----|----------|--|--|

To specify/override items for a sort of force quantities:

|     |       |     |    |    |    |    |          |  |  |
|-----|-------|-----|----|----|----|----|----------|--|--|
| DTI | INDTA | "2" | B1 | C1 | B2 | C2 | "ENDREC" |  |  |
|-----|-------|-----|----|----|----|----|----------|--|--|

**Examples:**

|     |       |   |  |  |  |  |  |  |  |
|-----|-------|---|--|--|--|--|--|--|--|
| DTI | INDTA | 0 |  |  |  |  |  |  |  |
|-----|-------|---|--|--|--|--|--|--|--|

To specify/override items for a sort of stress quantities:

|     |       |   |    |    |    |    |        |  |  |
|-----|-------|---|----|----|----|----|--------|--|--|
| DTI | INDTA | 1 | 64 | 18 | 75 | 18 | ENDREC |  |  |
|-----|-------|---|----|----|----|----|--------|--|--|

To specify/override items for a sort of force quantities:

|     |       |   |    |   |   |   |        |  |  |
|-----|-------|---|----|---|---|---|--------|--|--|
| DTI | INDTA | 2 | 34 | 2 | 2 | 4 | ENDREC |  |  |
|-----|-------|---|----|---|---|---|--------|--|--|

| Descriptor | Meaning   |
|------------|---|
| Bi         | Element type identification number. See the table in <a href="#">Item Codes, 1045</a> for allowable values. (Integer > 0)   |
| Ci         | Item code identification number for the stress, strain, or force quantity on which the sort or filter is to be performed. See the table in the <a href="#">Item Codes, 1045</a> for allowable values. (Integer) |

**Remarks:**

1. This table is recognized only in SOLs 101, 103, 105, 106, 108, 109, 111, 112, 114, 115, 144, 153, and for stress quantities only. One or more of the user parameters S1, S1G, or S1M must be specified with a value greater than or equal to zero in order to request sorting and/or filtering. See user parameter S1 in [Parameters, 783](#). In order to sort force or strain quantities, a DMAP Alter is required.
2. If the Ci value is -1, the element type will be suppressed on the output file. An example of this feature could be as follows: If an element type is to be sorted on two different values and output twice, this can be accomplished by two calls to the STRSORT module with two unique DTI tables. However, other element types will be printed twice. This additional print can be suppressed by setting their sort codes to -1.

3. [Table 13](#) lists the elements currently that are sortable. In addition, the element type identification number, the default stress output quantity, and the associated stress code identification numbers are provided. If this entry is not specified, then the stresses are sorted based on the default quantity given in [Table 13](#).

The following should be noted:

- a. The element type identification number is used internally by the program to differentiate element types.
- b. The stress code identification number is merely the word number in the standard printed output for the stress quantity of interest. For example, the thirteenth word of stress output for the CHEXA element is the octahedral shear stress. For this element type, the element identification number and the grid point ID each count as a separate word. Stress codes for the elements are tabulated in [Item Codes, 1045](#).
- c. By default, stress sorting for the membrane and plate elements will be performed on the Hencky-von Mises stress. For maximum shear stress, the STRESS (MAXS) Case Control command should be specified.

Table 13 Sortable Elements

| Element | Element Type ID Number | Default Stress Output Quantity and Identification Number |                       |
|---------|------------------------|--|-----------------------|
|         |                        | Quantity   | Stress Code ID Number |
| CBAR    | 34                     | Maximum stress at end B                                  | 14                    |
| CBEAM   | 2                      | Maximum stress at end B                                  | 108                   |
| CBEND   | 69                     | Maximum stress at end B                                  | 20                    |
| CONROD  | 10                     | Axial stress   | 2                     |
| CELAS1  | 11                     | Stress   | 2                     |
| CELAS2  | 12                     | Stress   | 2                     |
| CELAS3  | 13                     | Stress   | 2                     |
| CHEXA   | 67                     | Hencky-von Mises or Octahedral stress                    | 13                    |
| CQUAD4  | 33                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 17                    |
| CQUAD4* | 144                    | Maximum shear or Hencky-von Mises stress at $Z_2$        | 19                    |
| CQUAD8  | 64                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 19                    |
| CQUADR  | 82                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 19                    |
| CPENTA  | 68                     | Octahedral stress  | 13                    |
| CROD    | 1                      | Axial stress   | 2                     |
| CSHEAR  | 4                      | No default   | ---                   |
| CTETRA  | 39                     | No default   | ---                   |
| CTRIA3  | 74                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 17                    |
| CTRIA6  | 75                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 19                    |
| CTRIAR  | 70                     | Maximum shear or Hencky-von Mises stress at $Z_2$        | 19                    |
| CTRIAX6 | 53                     | No default   | ---                   |
| CTUBE   | 3                      | Axial stress   | 2                     |

\*CORNER output

**DTI,SETREE****Superelement Tree Definition**

Defines a superelement tree that determines the superelement processing order.

**Format:**

| 1   | 2      | 3       | 4     | 5       | 6      | 7       | 8     | 9       | 10 |
|-----|--------|---------|-------|---------|--------|---------|-------|---------|----|
| DTI | SETREE | "1"     | SEUP1 | SEDOWN1 | SEUP2  | SEDOWN2 | SEUP3 | SEDOWN3 |    |
|     | SEUP4  | SEDOWN4 | SEUP5 | SEDOWN5 | -etc.- |         |       |         |    |

**Example:**

|     |        |    |    |    |   |    |   |    |  |
|-----|--------|----|----|----|---|----|---|----|--|
| DTI | SETREE | 1  | 1  | 14 | 2 | 14 | 3 | 14 |  |
|     | 4      | 14 | 14 | 0  |   |    |   |    |  |

| Descriptor | Meaning  |
|------------|--|
| SEUPI      | Identification number of the superelement upstream from SEDOWNi. (Integer > 0)               |
| SEDOWNi    | Identification number of the superelement into which SEUPI is assembled. (Integer $\geq 0$ ) |

**Remarks:**

1. SETREE entries or the DTI,SETREE entry are required for multi-level superelement configurations.
2. If an DTI,SETREE entry is provided, then SETREE entries are not required.
3. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
4. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one or more of the superelements are not in the same path.
5. If this entry is not present, the superelement tree and the processing order are determined automatically.
6. A superelement identification may appear only once in a SEUPI field.
7. On restart, if a superelement identification does not appear in a SEUPI field, its matrices will not be assembled, even though they may be present in the database.
8. See the *MSC Nastran Superelements and Modules User's Guide* for a description of user-designated trees.

9. This entry is stored in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.
10. In the example above, the following superelement tree is defined:

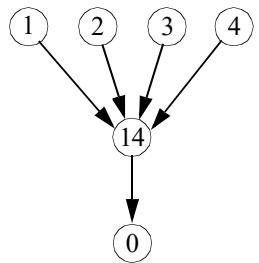


Figure 9-89 Sample Superelement Tree

**DTI,SPECSEL****Response Spectra Input Correlation Table**

Correlates spectra lines specified on TABLED1 entries with damping values.

**Format:**

| 1   | 2       | 3     | 4    | 5     | 6    | 7     | 8      | 9     | 10 |
|-----|---------|-------|------|-------|------|-------|--------|-------|----|
| DTI | SPECSEL | RECNO |      | TYPE  | TID1 | DAMP1 | TID2   | DAMP2 |    |
|     | TID3    | DAMP3 | TID4 | DAMP4 | TID5 | DAMP5 | -etc.- |       |    |

**Example:**

|     |         |   |     |   |   |     |   |     |  |
|-----|---------|---|-----|---|---|-----|---|-----|--|
| DTI | SPECSEL | 1 |     | A | 1 | .02 | 2 | .04 |  |
|     |         | 3 | .06 |   |   |     |   |     |  |
| DTI | SPECSEL | 3 |     | V | 4 | .01 |   |     |  |

| Descriptor        | Meaning   |
|-------------------|---|
| RECNO             | Spectrum number. (Integer > 0)  |
| TYPE              | Type of spectrum. (Character: "A" for acceleration, "V" for velocity, or "D" for displacement.) |
| TID <sub>i</sub>  | TABLED1 entry identification number. (Integer > 0)  |
| DAMP <sub>i</sub> | Damping value assigned to TID <sub>i</sub> . (Real)   |

**Remarks:**

1. The RECNO is the number of the spectrum defined by this entry. It is referenced on DLOAD Bulk Data entries.
2. The TID<sub>i</sub>, DAMP<sub>i</sub> pairs list the TABLED1 entry, which defines a line of the spectrum and the damping value assigned to it. The damping value is in the units of fraction of critical damping.
3. This entry is placed in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

## DTI,SPSEL

## Response Spectra Generation Correlation Table

Correlates output requests with frequency and damping ranges.

Format:

| 1   | 2     | 3     | 4     | 5      | 6  | 7  | 8  | 9  | 10 |
|-----|-------|-------|-------|--------|----|----|----|----|----|
| DTI | SPSEL | RECNO | DAMPL | FREQL  | G1 | G2 | G3 | G4 |    |
|     | G5    | G6    | G7    | -etc.- |    |    |    |    |    |

Example:

|     |       |    |   |   |    |    |    |    |  |
|-----|-------|----|---|---|----|----|----|----|--|
| DTI | SPSEL | 1  | 2 | 1 | 11 | 12 |    |    |  |
| DTI | SPSEL | 2  | 4 | 3 | 1  | 7  | 11 | 12 |  |
|     | 13    | 14 |   |   |    |    |    |    |  |

| Descriptor | Meaning   |
|------------|---|
| DAMPL      | Identification number of the FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values. (Integer > 0) |
| FREQL      | Identification number of the FREQi Bulk Data entry that specifies the list of frequencies. (Integer > 0)                    |
| Gi         | Grid point number where response spectra will be calculated. (Integer > 0)  |
| RECNO      | Record number of spectra to be generated. (Sequential integer beginning with 1.)  |

Remarks:

1. This table is used in SOLs 109 and 112.
2. Damping values are in the units of fraction of critical damping.
3. Output of response spectra requires the use of the XY PLOT...SPECTRA(RECNO)/Gi... command, where Gi is restricted to the grid points listed on the (RECNO) record of this entry.
4. The SPSEL table is stored in the database automatically in SOLs 109 and 112. Once stored, the Bulk Data entry may be removed from the input file.
5. There must be case control output request for displacements and velocities of the points to be output. For example you could use case control commands:

DISP(PLOT) = ALL

VELO(PLOT) = ALL

## DTI,UNITS

## Unit Definitions

Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface or a Nastran fatigue analysis.

Format:

| 1   | 2     | 3 | 4    | 5     | 6      | 7    | 8      | 9 | 10 |
|-----|-------|---|------|-------|--------|------|--------|---|----|
| DTI | UNITS | 1 | MASS | FORCE | LENGTH | TIME | STRESS |   |    |

Example:

|     |       |   |    |   |   |   |     |  |  |
|-----|-------|---|----|---|---|---|-----|--|--|
| DTI | UNITS | 1 | KG | N | M | S | MPA |  |  |
|-----|-------|---|----|---|---|---|-----|--|--|

Remarks:

1. The DTI,UNITS Bulk Data entry is required for a ADAMSMNF FLEXBODY=YES run. See the [ADAMSMNF\\* \(Case\)](#), 222 case control entry. ADAMS is not a unitless code (as is Nastran). Units must be specified. A DTI Bulk Data entry provides 'UNITS' (a unique identifier) input as the above example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed in the table below. MASS, FORCE, LENGTH, and TIME are required for ADAMS interface.
2. MSC Nastran is a unitless code and it is the user's responsibility to ensure compatible units. During a fatigue analysis, stress is converted to SI units of MPa because the fatigue material property stress parameters as defined on the MATFTG entry are internally converted to standard SI units of MPa. The stresses from the analysis must match. Thus it is necessary for the user to use DTI,UNITS to define the stress units to ensure proper conversion. The default units are MPa and this entry is only necessary if stresses are not in MPa.

| Mass             | Force                 | Length          | Time             | Stress*                          |
|------------------|-----------------------|-----------------|------------------|----------------------------------|
| KG - kilogram    | N - newton            | KM - kilometer  | H - hour         | MPA - megapascal                 |
| LBM - pound-mass | LBF - pounds-force    | M - meter       | MIN - minute     | PA - pascal                      |
| SLUG - slug      | KGF - kilograms-force | CM - centimeter | S - second       | PSI - pound per square inch      |
| GRAM - gram      | OZF - ounce-force     | MM - millimeter | MS - millisecond | KSI - kilo pound per square inch |
| OZM - ounce-mass | DYNE - dyne           | MI - mile       | US - microsecond | PSF - pound per square foot      |

| Mass                              | Force                              | Length           | Time                 | Stress*                              |
|-----------------------------------|------------------------------------|------------------|----------------------|--------------------------------------|
| KLBM - kilo pound-mass (1000 lbm) | KN - kilonewton                    | FT - foot        | NANOSEC - nanosecond | KSF - kilo pound per square foot     |
| MGG - megagram                    | KLBF - kilo pound-force (1000 lbf) | IN- inch         | D - day              | DYNECM2 - dyne per square centimeter |
| SLINCH - 12 slugs                 | MN - millinewton                   | UM - micrometer  |                      | BAR - bar                            |
| UG - microgram                    | UN - micronewton                   | NM - nanometer   |                      | ATM - physical atmosphere            |
| NG - nanogram                     | NN - nanonewton                    | ANG - angstrom   |                      |                                      |
| USTON - US ton                    |                                    | YD - yard        |                      |                                      |
|                                   |                                    | MIL - milli-inch |                      |                                      |
|                                   |                                    | UIN - micro-inch |                      |                                      |

\* For fatigue analysis of spot welds, only MPA, PA, PSI, KSI, PSF, and KSF are supported.

For random vibration fatigue analysis using SOL 108 or 111, only MPA, PA, PSI, and KSI are supported.

**DVBSHAP****Design Variable to Boundary Shapes**

Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.

**Format:**

| 1       | 2    | 3      | 4    | 5   | 6    | 7   | 8    | 9   | 10 |
|---------|------|--------|------|-----|------|-----|------|-----|----|
| DVBSHAP | DVID | AUXMOD | COL1 | SF1 | COL2 | SF2 | COL3 | SF3 |    |

**Example:**

|         |   |   |   |     |  |  |  |  |  |
|---------|---|---|---|-----|--|--|--|--|--|
| DVBSHAP | 4 | 1 | 1 | 1.6 |  |  |  |  |  |
|---------|---|---|---|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| DVID       | Design variable identification number of a DESVAR entry. (Integer > 0)                   |
| AUXMOD     | Auxiliary model identification number. (Integer > 0)                                     |
| COLi       | Load sequence identification number from AUXMODEL Case Control command.<br>(Integer > 0) |
| SFi        | Scaling factor for load sequence identification number. (Real; Default = 1.0)            |

**Remarks:**

1. Design variable DVID must be defined on a DESVAR entry.
2. Multiple references to the same DVID and/or COLi will result in the vector addition of the referenced boundary shape vectors.
3. Multiple DVBSHAP entries may be specified.

**DVCREL1****Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables.

**Format:**

| 1       | 2     | 3     | 4     | 5      | 6     | 7     | 8      | 9 | 10 |
|---------|-------|-------|-------|--------|-------|-------|--------|---|----|
| DVCREL1 | ID    | TYPE  | EID   | CPNAME | CPMIN | CPMAX | C0     |   |    |
|         | DVID1 | COEF1 | DVID2 | COEF2  | DVID3 | COEF3 | -etc.- |   |    |

**Example:**

|         |   |        |   |       |  |     |  |  |  |
|---------|---|--------|---|-------|--|-----|--|--|--|
| DVCREL1 | 5 | CQUAD4 | 1 | ZOFFS |  | 1.0 |  |  |  |
|         | 1 | 1.0    |   |       |  |     |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number. (Integer > 0)   |
| TYPE       | Name of an element connectivity entry, such as "CBAR", "CQUAD4", etc. (Character)   |
| EID        | Element Identification number. (Integer > 0)  |
| CPNAME     | Name of connectivity property, such as "X1", "X2", "X3", "ZOFFS", etc. (Character)  |
| CPMIN      | Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real) |
| CPMAX      | Maximum value allowed for this property. See Remark 4. (Real; Default =-1.0E+20)  |
| C0         | Constant term of relation. (Real; Default = 0.0)  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)   |
| COEFi      | Coefficient of linear relation or keyword = "PVAL". (If i = 1, Real or Character; if i > 1, Real)   |

**Remarks:**

1. The relationship between the connectivity property and design variables is given by:

$$CP_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc), ZOFFS (case insensitive) must be specified on the CPNAME field.

4. The default values for CPMIN and CPMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the associated DESVAR entry will keep the designed property within meaningful bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the connectivity bulk data entry. If a DVCREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
6. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL1 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL1 entry should refer to CQUAD4/CTRIA3 types.

**DVCREL2****Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables with a user-supplied equation.

**Format:**

| 1       | 2        | 3     | 4      | 5      | 6     | 7     | 8     | 9     | 10 |
|---------|----------|-------|--------|--------|-------|-------|-------|-------|----|
| DVCREL2 | ID       | TYPE  | EID    | CPNAME | CPMIN | CPMAX | EQID  |       |    |
|         | "DESVAR" | DVID1 | DVID2  | DVID3  | DVID4 | DVID5 | DVID6 | DVID7 |    |
|         |          | DVID8 | -etc.- |        |       |       |       |       |    |
|         | "DTABLE" | LABL1 | LABL2  | LABL3  | LABL4 | LABL5 | LABL6 | LABL7 |    |
|         |          | LABL8 | -etc.- |        |       |       |       |       |    |

**Example:**

|         |        |      |     |    |      |     |     |  |  |
|---------|--------|------|-----|----|------|-----|-----|--|--|
| DVCREL2 | 1      | CBAR | 100 | X1 | 0.05 | 1.0 | 100 |  |  |
|         | DESVAR | 1001 |     |    |      |     |     |  |  |
|         | DTABLE | X10  |     |    |      |     |     |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number. (Integer > 0)   |
| TYPE       | Name of an element connectivity entry, such as "CBAR", "CQUAD4", etc. (Character)   |
| EID        | Element Identification number. (Integer > 0)  |
| CPNAME     | Name of connectivity property, such as "X1", "X2", "X3", "ZOFFS", etc. (Character)  |
| CPMIN      | Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real) |
| CPMAX      | Maximum value allowed for this property. (Real; Default =1.0E+20)   |
| EQID       | DEQATN entry identification number. (Integer > 0)   |
| "DESVAR"   | DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)   |
| DVIDi      | DESVAR entry identification number. (Integer > 0)   |
| "DTABLE"   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)       |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)  |

**Remarks:**

1. The variable identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where N = m + n) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc.), ZOFFS (case insensitive) must be specified on the CPNAME field.
4. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL2 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL2 entry should refer to CQUAD4/CTRIA3 types.

**DVGRID**

## Design Variable to Grid Point Relation

Defines the relationship between design variables and grid point locations.

Format:

| 1      | 2    | 3   | 4   | 5     | 6  | 7  | 8  | 9 | 10 |
|--------|------|-----|-----|-------|----|----|----|---|----|
| DVGRID | DVID | GID | CID | COEFF | N1 | N2 | N3 |   |    |

Example:

|        |   |     |   |     |     |     |     |  |  |
|--------|---|-----|---|-----|-----|-----|-----|--|--|
| DVGRID | 3 | 108 | 5 | 0.2 | 0.5 | 0.3 | 1.0 |  |  |
|--------|---|-----|---|-----|-----|-----|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| DVID       | DESVAR entry identification number. (Integer > 0)  |
| GID        | Grid point (GRID) or geometric point (POINT) identification number. (Integer > 0)                              |
| CID        | Coordinate system identification number. (Integer $\geq 0$ ; Default = 0)                                      |
| COEFF      | Multiplier of the vector defined by Ni. (Real; Default = 0.0)  |
| Ni         | Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$ ) |

Remarks:

1. A CID of zero or blank (the default) references the basic coordinate system.
2. Multiple references to the same grid ID and design variable result in vectorial addition of the participation vectors defined by CID, COEFF, and Ni. There is no restriction on the number of DVGRID entries that may reference a given grid (GID) or design variable (DVID).
3. The coordinate update equation is given as

$$\{g\}_i - \{g\}_i^0 = \sum_j COEFF_j (X_{DVIDj} - X_{DVIDj}^0) \{N\}_j$$

where  $\{g\}_i$  is the location of the  $i$ -th grid,  $[g_x g_y g_z]^T$ .

The vector  $\{N\} = [N_x N_y N_z]^T$  is determined from CID and Ni. Note that it is a change in a design variable from its initial value  $X^0$ , and not the absolute value of the design variable itself, that represents a change in a grid point location,  $\{g\}_i - \{g\}_i^0$ .

4. The DVGRID entry defines the participation coefficients (basis vectors) of each design variable for each of the coordinates affected by the design process in the relationship

$$\{\Delta g\}_i = \sum_j \{T\}_{ij} \cdot \Delta X_j$$

5. DVGRID entries that reference grid points on MPCs or RSSCON entries produce incorrect sensitivities. Often the sensitivities are 0.0 which may result in a warning message indicating zero gradients which may be followed by UFM 6499. Other rigid elements produce correct results.

**DVLREL1**

Defines the linear relation between analysis model loading and design variables in SOL 200 with Analysis = STATICS

Format:

| 1       | 2    | 3    | 4     | 5     | 6     | 7    | 8  | 9 | 10 |
|---------|------|------|-------|-------|-------|------|----|---|----|
| DVLREL1 | ID   | TYPE | SID   | LNAME | LMIN  | LMAX | C0 |   |    |
|         | ATT1 | ATT2 | ATT3  | ATT4  | ATT5  |      |    |   |    |
|         | DVID | COEF | DVID2 | COEF2 | DVID3 | Etc  |    |   |    |

Example: Design the N1 FORCE at GRID 100 to be equal to DESVAR=10

|         |     |       |     |    |  |  |  |  |  |
|---------|-----|-------|-----|----|--|--|--|--|--|
| DVLREL1 | 10  | FORCE | 300 | N1 |  |  |  |  |  |
|         | 100 |       |     |    |  |  |  |  |  |
|         | 10  | 1.0   |     |    |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number (Integer>0)   |
| TYPE       | Name of Load, such as FORCE, see Remark 2. for supported types (Character)                                 |
| SID        | Load set ID (Integer>0)  |
| LNAME      | Load Name, such as F or N1 on the FORCE entry. See Remark 2. (Character)                                   |
| LMIN       | Minimum value for the load. See Remark 5. (Real, default=-1.0e35)  |
| LMAX       | Maximum value for the load (Real, Default=1.0e20)  |
| C 0        | Constant term of relation (Real, Default=0.0)  |
| ATTi       | Attributes of the designed load, see Remark 2. (Integer>0 or blank)  |
| DVIDi      | DESVAR entry identification number. (Integer>0)  |
| COEFFi     | Coefficient of linear relation or keyword="PVAL", See Remark 3. (If i=1, Real or Character; if i>1, Real ) |

Remarks:

1. The relationship between the analysis model load and the design variables is given by:

$$L_i = C_0 + \sum COEFF_i X_{DVID_i}$$

2. Supported TYPES and their ATT<sub>i</sub> meaning are given in the following table:

| Load Type | LNAME<br>(character) | ATT1<br>(integer)                   | ATT2<br>(integer)     | ATT3<br>(integer) | ATT4<br>(integer) | ATT5<br>(integer) |
|-----------|----------------------|-------------------------------------|-----------------------|-------------------|-------------------|-------------------|
| FORCE     | F, N1, N2 or N3      | G                                   | CID<br>(see remark 6) |                   |                   |                   |
| LOAD      | S or Si              | 0 if LNAME<br>is S, Li<br>otherwise |                       |                   |                   |                   |
| MOMENT    | M, N1, N2 or<br>N3   | G                                   | CID (see<br>remark 6) |                   |                   |                   |

3. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the load bulk data entry. If a DVLREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
4. If there are multiple loads that satisfy the designation of the request (for example two FORCE entries with identical SID, GRID and CID values), it is a user input error.
5. The default values of LMIN and LMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds
6. Input of these data is optional. In most cases, leaving it off will result in a single load that qualifies with the remaining attributes. If there are multiple instances that qualify, it is a user input error.
7. The use of this entry is limited to statics and buckling analyses in SOL 200.

**DVMREL1****Design Variable to Material Relation**

Defines the relation between a material property and design variables.

**Format:**

| 1       | 2     | 3     | 4     | 5      | 6     | 7     | 8      | 9 | 10 |
|---------|-------|-------|-------|--------|-------|-------|--------|---|----|
| DVMREL1 | ID    | TYPE  | MID   | MPNAME | MPMIN | MPMAX | C0     |   |    |
|         | DVID1 | COEF1 | DVID2 | COEF2  | DVID3 | COEF3 | -etc.- |   |    |

**Example:**

|         |   |      |   |     |      |     |  |  |  |
|---------|---|------|---|-----|------|-----|--|--|--|
| DVMREL1 | 5 | MAT1 | 1 | RHO | 0.05 | 1.0 |  |  |  |
|         | 1 | 1.0  |   |     |      |     |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)  |
| TYPE       | Name of a material property entry, such as "MAT1", "MAT2", etc. (Character)  |
| MID        | Material Identification number. (Integer > 0)  |
| MPNAME     | Name of material property, such as "E" or "RHO". (Character)   |
| MPMIN      | Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real) |
| MPMAX      | Maximum value allowed for this property. See Remark 4. (Real; Default = 1.0E+20)   |
| C0         | Constant term of relation. (Real; Default = 0.0)   |
| DVIDi      | DESVAR entry identification number. (Integer > 0)  |
| COEFi      | Coefficient of linear relation or keyword = "PVAL". (If i = 1, Real or Character; if i > 1, Real)  |

**Remarks:**

1. The relationship between the material property and design variables is given by:

$$MP_i = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries, 1117](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

4. The default value for MPMIN and MPMAX are not applied when the linear property is a function of a single design variable and C0=0.0. It is expected that the limits applied to the DESVAR entry will keep the designed property within reasonable bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the material bulk data entry. If a DVMREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.

**DVMREL2****Design Variable to Material Relation**

Defines the relation between a material property and design variables with a user-supplied equation.

**Format:**

| 1       | 2      | 3     | 4      | 5      | 6     | 7     | 8     | 9     | 10 |
|---------|--------|-------|--------|--------|-------|-------|-------|-------|----|
| DVMREL2 | ID     | TYPE  | MID    | MPNAME | MPMIN | MPMAX | EQID  |       |    |
|         | DESVAR | DVID1 | DVID2  | DVID3  | DVID4 | DVID5 | DVID6 | DVID7 |    |
|         |        | DVID8 | -etc.- |        |       |       |       |       |    |
|         | DTABLE | LABL1 | LABL2  | LABL3  | LABL4 | LABL5 | LABL6 | LABL7 |    |
|         |        | LABL8 | -etc.- |        |       |       |       |       |    |

**Example:**

|         |        |      |   |   |      |     |     |  |  |
|---------|--------|------|---|---|------|-----|-----|--|--|
| DVMREL2 | 5      | MAT1 | 1 | E | 0.05 | 1.0 | 100 |  |  |
|         | DESVAR | 1    | 2 |   |      |     |     |  |  |
|         | DTABLE | E0   |   |   |      |     |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)  |
| TYPE       | Name of a material property entry, such as "MAT1", "MAT2", etc. (Character)  |
| MID        | Material Identification number. (Integer > 0)  |
| MPNAME     | Name of material property, such as "E" or "RHO". (Character)   |
| MPMIN      | Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real) |
| MPMAX      | Maximum value allowed for this property. (Real; Default = 1.0E+20)   |
| EQID       | DEQATN entry identification number. (Integer > 0)  |
| DESVAR     | DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)  |
| DTABLE     | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)    |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)   |

**Remarks:**

1. The variables identified by DVID<sub>i</sub> and LABL<sub>i</sub> correspond to variable names (x<sub>1</sub>, x<sub>2</sub>, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x<sub>1</sub> through x<sub>N</sub> (where N = m + n) are assigned in the order DVID<sub>1</sub>, DVID<sub>2</sub>, ..., DVID<sub>m</sub>, LABL<sub>1</sub>, LABL<sub>2</sub>, ..., LABL<sub>n</sub>.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries, 1117](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

**DVPREL1****Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables.

**Format:**

| 1       | 2     | 3     | 4     | 5             | 6     | 7      | 8  | 9 | 10 |
|---------|-------|-------|-------|---------------|-------|--------|----|---|----|
| DVPREL1 | ID    | TYPE  | PID   | PNAME/<br>FID | PMIN  | PMAX   | C0 |   |    |
|         | DVID1 | COEF1 | DVID2 | COEF2         | DVID3 | -etc.- |    |   |    |

**Example:**

|         |    |      |     |      |     |     |  |  |  |
|---------|----|------|-----|------|-----|-----|--|--|--|
| DVPREL1 | 12 | PBAR | 612 | 6    | 0.2 | 3.0 |  |  |  |
|         | 4  | 0.25 | 20  | 20.0 | 5   | 0.3 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)  |
| TYPE       | Name of a property entry, such as "PBAR", "PBEAM", etc. (Character)  |
| PID        | Property entry identification number. (Integer > 0)  |
| PNAME/FID  | Property name, such as "T", "A", or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0) |
| PMIN       | Minimum value allowed for this property. If PMIN references a property that can only be positive, then the default value for PMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 6. (Real)   |
| PMAX       | Maximum value allowed for this property. (Real; Default = 1.0E+20)   |
| C0         | Constant term of relation. (Real; Default = 0.0)   |
| DVIDi      | DESVAR entry identification number. (Integer > 0)  |
| COEFi      | Coefficient of linear relation or keyword = "PVAL". See Remark 7. (If i = 1, Real or Character; if i > 1, Real)  |

**Remarks:**

1. The relationship between the analysis model property and design variables is given by:

$$P_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. TYPE="PBEND" is not supported. TYPE="PBEAML" supports only PNAME and not FID.

4. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either PNAME=A, FID=+4 or FID=-3 can be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
5. Designing PBEAML or PBEAM requires specification of both property name and station. [Table 14](#) shows several examples.

**Table 14**

| PTYPE  | Property Name | END A              | END B   | i-th Station |
|--------|---------------|--------------------|---------|--------------|
| PBEAML | DIM1          | DIM1 or<br>DIM1(A) | DIM1(B) | DIM1(i)      |
| PBEAM  | A             | A or A(A)          | A(B)    | A(i)         |

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, referencing an END B property name on a DVPREL1 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

6. The default values of PMIN and PMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds.
7. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the property bulk data entry. If a DVPREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
8. With GPLY for TYPE field and GPLYID for PID field, a ply identified with GPLYID across all PCOMPG entries in the model can be designed. Internally, a DVPREL1 will be spawned for each PCOMPG has a ply ID of GPLYID. For TYPE=GPLY, the relationship between the analysis model property and design variables is given by.

$$P_i = C0 + (T0_i \text{ or } THETA0_i) \cdot \sum_j (COEF_j \cdot X_{DVID_j}) \text{ for PNAME=T or THETA}$$

Where T0 and THETA0 are value of thickness and theta angle on the original PCOMPG. Note that non-zero C0 is not recommended for TYPE=GPLY or PCOMPG. For THETA0 with original value equal to 0.0, THETA0 is taken as 1.0 and it is recommended to have XINIT of DVID set to 0.0.

**DVPREL2****Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables with a user-supplied equation.

**Format:**

| 1       | 2        | 3     | 4      | 5             | 6     | 7     | 8     | 9     | 10 |
|---------|----------|-------|--------|---------------|-------|-------|-------|-------|----|
| DVPREL2 | ID       | TYPE  | PID    | PNAME/<br>FID | PMIN  | PMAX  | EQID  |       |    |
|         | "DESVAR" | DVID1 | DVID2  | DVID3         | DVID4 | DVID5 | DVID6 | DVID7 |    |
|         |          | DVID8 | -etc.- |               |       |       |       |       |    |
|         | "DTABLE" | LABL1 | LABL2  | LABL3         | LABL4 | LABL5 | LABL6 | LABL7 |    |
|         |          | LABL8 | -etc.- |               |       |       |       |       |    |

**Example:**

|         |        |      |     |    |     |     |    |  |  |
|---------|--------|------|-----|----|-----|-----|----|--|--|
| DVPREL2 | 13     | PBAR | 712 | 5  | 0.2 | 0.4 | 50 |  |  |
|         | DESVAR | 4    | 11  | 13 | 5   |     |    |  |  |
|         | DTABLE | PI   | YM  |    |     |     |    |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)  |
| TYPE       | Name of a property entry, such as PBAR, PBEAM, etc. (Character)  |
| PID        | Property entry identification number. (Integer > 0)  |
| PNAME/FID  | Property name, such as "T", "A", or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)                         |
| PMIN       | Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.E-15) |
| PMAX       | Maximum value allowed for this property. (Real; Default = 1.0E20)  |
| EQID       | DEQATN entry identification number. (Integer > 0)  |
| "DESVAR"   | DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)  |
| "DTABLE"   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)  |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)   |

**Remarks:**

1. The variables identified by DVID<sub>i</sub> and LABL<sub>i</sub> correspond to variable names (x<sub>1</sub>, x<sub>2</sub>, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x<sub>1</sub> through x<sub>N</sub> (where N = m+n) are assigned in the order DVID<sub>1</sub>, DVID<sub>2</sub>, ..., DVID<sub>n</sub>, LABL<sub>1</sub>, LABL<sub>2</sub>, ..., LABL<sub>m</sub>.
2. If both "DESVAR" and "DTABLE" are specified in field 2, "DESVAR" must appear first.
3. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either PNAME=A, FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
4. Types "PBEND", "PBTRL" and "PBEAML" are not supported for the DVPREL2.
5. Designing PBEAM requires specification of both property name and station. [Table 15](#) shows one example.

**Table 15**

| PTYPE | Property Name | END A     | END B | i-th Station |
|-------|---------------|-----------|-------|--------------|
| PBEAM | A             | A or A(A) | A(B)  | A(i)         |

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, referencing an END B property name on a DVPREL2 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

**DVSHAP****Design Variable to Basis Vector(s)**

Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.

**Format:**

| 1      | 2    | 3    | 4   | 5    | 6   | 7    | 8   | 9 | 10 |
|--------|------|------|-----|------|-----|------|-----|---|----|
| DVSHAP | DVID | COL1 | SF1 | COL2 | SF2 | COL3 | SF3 |   |    |

**Example:**

|        |   |   |     |   |     |  |  |  |  |
|--------|---|---|-----|---|-----|--|--|--|--|
| DVSHAP | 2 | 1 | 2.0 | 4 | 1.0 |  |  |  |  |
|--------|---|---|-----|---|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| DVID       | Design variable identification number on the DESVAR entry. (Integer > 0)  |
| COLi       | Column number of the displacement matrix. See Remark 2. ( $1 \leq \text{Integer} \leq$ maximum column number in the displacement matrix.) |
| SFi        | Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)  |

**Remarks:**

1. DVID must be defined on a DESVAR entry.
2. COLi must be a valid column number in the displacement matrix.
3. Multiple references to the same DVID and/or COLi will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the second column.
4. The displacement matrix must have been created by SOL 101 or 200 with analysis = statics and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN    DISPMAT=' physical filename of MASTER DBset '
DBLOCATE DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D) ,
LOGICAL=DISPMAT
```

**DVPSURF**

## Design Variable to Control Surface Setting Relation

Defines the relationship between a control surface setting in a particular subcase and a design variable.

**Format:**

| 1       | 2  | 3       | 4      | 5    | 6    | 7 | 8 | 9 | 10 |
|---------|----|---------|--------|------|------|---|---|---|----|
| DVPSURF | ID | AELABEL | TRIMID | DVID | COEF |   |   |   |    |

**Example:**

|         |    |         |   |     |         |  |  |  |  |
|---------|----|---------|---|-----|---------|--|--|--|--|
| DVPSURF | 10 | OBDFLAP | 1 | 100 | 0.01746 |  |  |  |  |
|---------|----|---------|---|-----|---------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number (Integer>0)                                 |
| AELABEL    | LABEL of the AESURF entry that is being designed (Character ,no default) |
| TRIMID     | Associated trim set identification number (Integer>0)                    |
| DVID       | DESVAR entry identification number (Integer>0)                           |
| COEF       | Coefficient of linear relation (Real)                                    |

**Remarks:**

1. The relationship between the control surface setting and the design variable is given by  $\delta_{\text{SURF}} = \text{COEF} * X_{\text{DVID}}$
2. The surface called out by AELABEL must also appear on the trim entry specified by TRIMID. The value specified on the trim entry will be overwritten by the value obtained from the relationship of Remark 1.
3. Limits on the control deflection are not provided on this entry but can be specified on the underlying DESVAR
4. The DVID called out on this entry cannot be associated with any other designed property.
5. Note that since the DVPSURF calls out a TRIM ID, it is associated only with a single subcase.

## DYFSISW

### Fluid Structure Interaction Activation Switch

Allows activating or deactivating Fluid Structure Interaction and Eulerian solver. Used in SOL 700 only.

Format:

| 1       | 2    | 3   | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|-----|-------|---|---|---|---|---|----|
| DYFSISW | SWID | TID | INITV |   |   |   |   |   |    |

Example:

|         |    |    |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|
| DYFSISW | 14 | 10 |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |    |                                   |     |   |
|------------|---|----|-----------------------------------|-----|---|
| SWID       | Unique number of a DYFSISW entry. (Integer > 0; Required)   |    |                                   |     |   |
| TID        | TABLED1 ID. Fluid Structure Interface and Euler solver are switched on and off, depending on the y-value of the table. The x-value of the table represents the time; the y-value denotes: <table> <tr> <td>ON</td><td><math>y &gt; 0.0</math></td></tr> <tr> <td>OFF</td><td><math>y &lt; 0.0</math></td></tr> </table> | ON | $y > 0.0$                         | OFF | $y < 0.0$   |
| ON         | $y > 0.0$   |    |                                   |     |   |
| OFF        | $y < 0.0$   |    |                                   |     |   |
| INITV      | Euler element initialization flag. See remark 3. (Character, Default=NO) <table> <tr> <td>NO</td><td>Element initialization at cycle 0</td></tr> <tr> <td>YES</td><td>Element initialization when the elements becomes active</td></tr> </table>  | NO | Element initialization at cycle 0 | YES | Element initialization when the elements becomes active |
| NO         | Element initialization at cycle 0   |    |                                   |     |   |
| YES        | Element initialization when the elements becomes active   |    |                                   |     |   |

Remarks:

1. The default is that COUPLE, AIRBAG and Euler solver is active at all times.
2. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
3. For simulations involving a prestress phase the Euler elements can be initially deactivated. When the Euler elements are activated the Eulerian masses still originate from the Euler initiation at cycle 0. If during prestressing the structure did not move much, then these Eulerian masses of cycle 0 can give a stable run. But if there has been substantial movement of the structure then Eulerian masses can be erroneously compressed. This shows up by large pressures and velocities in the Euler elements and a time step too small. To avoid this instability the Euler initialization can be postponed until the Euler elements become active.

**DYPARAM, ATBAOUT**

Output Frequency to Main Output File of ATB - SOL 700

Defines the frequency at which output is written to the main output file of ATB. Used in SOL700 only.

**Format:**

DYPARAM, ATBAOUT, value

**Example:**

DYPARAM, ATBAOUT, 5.0E-3

| Option | Content  |
|--------|--|
| Value  | Every multiple of ATBAOUT seconds, the main output file of ATB is updated. (Real > 0.0; default=10.0E-3) |

**Remarks:**

1. Only active when field 3 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of the output of segment acceleration, velocity and displacement, joint forces and moments.

## DYPARAM, ATBHOUT

Write ATB Output to Time history files - SOL 700

A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11.  
Used in SOL700 only.

Format:

DYPARAM, ATBHOUT, option

Example:

DYPARAM, ATBHOUT, NO

| Option | Content  |
|--------|--|
| Option | Flag of ATBHOUT. (Character; default=YES)<br><br>YES: The time history files are created.<br><br>NO: The time history files are not created. |

**DYPARAM, ATBTOUT**

Output Frequency to Time-History Files of ATB - SOL 700

Defines the frequency at which output is written to the time-history files of ATB. Used in SOL700 only.

**Format:**

DYPARAM, ATBTOUT, value

**Example:**

DYPARAM, ATBTOUT, 1.0E-4

| Option | Content   |
|--------|---|
| Value  | Every multiple of ATBTOUT seconds, the time-history files of ATB are updated.<br>(Real > 0.0; default=1.0E-3) |

**Remarks:**

1. Only active when field 26 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of all output requested on the H-cards, and of the tabular time-histories that are controlled by field 18 on the A5 card of the ATB input file.

**DYPARAM,AUTOCOUP****Automatic Coupling**

Defines the automatic coupling algorithm. Used in Sol700 only.

**Format:**

DYPARAM,AUTOCOUP,ACTIVE,CLEAN,DUMMY, TOL\_AREA, OUTPUT, FAIL

**Example:**

DYPARAM, AUTOCOUP, ON

| Descriptor | Meaning  |
|------------|--|
| ACTIVE     | Flag of activation of the sub element approach. (Character; default=OFF)   |
| ON         | Turn on the sub element approach.  |
| OFF        | Turn off the sub element approach.   |
| CLEAN      | Flag of cleaning obsolete sub elements. (Character; default=OFF)   |
| ON         | Cleans obsolete sub elements   |
| OFF        | Obsolete elements will not be removed from memory.   |
| DUMMY      | Flag of putting dummy segments on coupling surface archives. (Character; default=OFF)  |
| ON         | Puts dummy segments on the coupling surface archive  |
| OFF        | Does not put dummy segments on the coupling surface archive.   |
| TOL_AREA   | If the area of a hole is smaller than TOL_AREA a 1D flow computation method is used.<br>(Real $\geq 0.0$ ; default=0.0).   |
| OUTPUT     | Determines how results of Euler elements that are intersected by the structure are written to the Euler archive. (Character, Default = ZERO).                        |
| ZERO       | Write zero for the results of these intersected elements.  |
| AVERAGE    | Average across sub elements. Each sub element has the same weight.   |
| AVEREAGEU  | Average across sub element using the uncover fraction as weights.  |
| MAXUNC     | The results are taken from the sub element with the largest volume uncover fraction.   |
| COVER      | Uses the cover field of COUPLE to select the proper sub element. Only supported for COVER=OUTSIDE or COVER=INSIDE. Not supported for COVER =NONE.                    |
| FAIL       | Activates fluid flow through failed shell segments in the coupling surface. (Character; there is no default, if failure is not used, then FAIL should be left blank) |
| POROUS     | Makes failed segment porous. This is the fast coupling approach.   |

REMOVE      Removes the failed segments from the coupling surface.

**Remarks:**

1. Features that are not supported are:
  - Multiple Euler domain
  - Adaptive Euler meshes
  - Fluid flow through failed segments in the coupling surface failure can only be defined by option FAIL of DYPARAM, AUTOCOUP. Defining this failure by use of COUP1FL and COUPINT is not supported.
  - COUPLE with option AIRBAG
  - Graded meshes
  - Euler import
  - Viscosity
  - Porosity
  - Option FAIL=POROUS and REMOVE are not supported by DMP.
  - Markers
2. With automatic coupling fluid can be defined on both sides of the coupling surface. To do this option COVER of the COUPLE entry has to be set to NONE. When using COVER = OUTSIDE fluid is only initialized in the inside region of the coupling surface. For COVER = INSIDE the opposite applies. When the coupling surface is not closed, only COVER=NONE is allowed.
3. In the auto coupling approach holes in the coupling surface are meshed with dummy segments. For holes with large deformations PARAM DUMSEGS, ON can be used to maintain good dummy segments.
4. For adding FAIL = POROUS failed segments have to be made fully porous. This is straightforward.
5. FAIL = REMOVE. When the structure fails, holes in the coupling surface are formed. For a closed coupling surface each side of a segment is connected to another segment. Therefore, the segments have no free edges. But a segment adjacent to a hole has a free edge. At this edge it is not connected to any other segment. A list of free edges of the coupling surface specifies the holes. By maintaining a list of free edges auto coupling handles these holes automatically. If a segment fails, this list of free edges has to be updated. Also the failed segment has to be removed from the coupling surface.

Note:

This entry also supports DMP. While using for DMP, no changes to input is needed. In the auto coupling method Euler elements that are intersected by the structure are split into sub elements. For details refer to [1]. To enable dmp for auto coupling some of the properties of sub elements have to be communicated across CPU's. If holes in the coupling surface extend across multiple CPUs, also communication across CPU's is needed.

**DYPARAM,AXIALSYM**

## Axial Symmetric Analyses - SOL 700

Enables an efficient and accurate 2-D axial symmetry for Eulerian materials. A much larger time step becomes possible by not taking into account the mesh-size in circumferential direction. Used in SOL 700 only.

**Format:**

DYPARAM,AXIALSYM,MESHTYPE,AXIALAXIS,SYMPLAN,PHI,ALIGN,PHI2

**Example:**

DYPARAM,AXIALSYM,RECT,X,XY,2.5,YES,0.0

| Descriptor | Meaning   |                              |
|------------|---|------------------------------|
| MESHTYPE   | Two types of Euler meshes are supported: (Character; Required)  |                              |
| AXIAL      | Axial symmetric meshes.   |                              |
| RECT       | Rectangular meshes  |                              |
| AXIAL AXIS | X   | X-Axis (Character; Required) |
|            | Y   | Y-Axis                       |
|            | Z   | Z-Axis                       |
| SYMPLAN    | The approximate symmetry plan of the Euler mesh. On for MESHTYPE=AXIAL. See Remark 7. (Character; Required)                           |                              |
| XY         | XY-Plane  |                              |
| YZ         | YZ-Plane  |                              |
| ZX         | ZX-Plane  |                              |
| PHI        | Only used for MESHTYPE = RECT. Used to create a 2D axial symmetric mesh with angles +PHI/2 and -PHI/2. (Real; Default = 0.0)          |                              |
| ALIGN      | Only used for MESHTYPE = AXIAL. (Character; Default = YES)  |                              |
| YES        | Align normals of oblique Euler element faces. This prevents errors in strains that can arise from small errors in Euler face normals. |                              |
| NO         | Do not align normals.   |                              |
| PHI2       | As a final operation rotate the mesh around the axial axis by the angle PHI2. See Remark 6. (Real; Default = 0.0)                     |                              |

**Remarks:**

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.

2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one layer can be used. Using the angle specified by PHI this Euler mesh is mapped into a 2d axial symmetric mesh.
  3. The Euler mesh has to consist of one layer.
  4. Rectangular meshes that can be made 2d symmetric using the angle PHI should satisfy:
    - All boundary Euler faces are aligned with a coordinate direction
    - Only one layer thick.
    - The axial symmetry axis is either on the boundary of the Euler mesh or outside the Euler mesh. It is not allowed that the axial axis is inside the Euler mesh.
- Initialization of Euler element using geometric regions as defined by the TICEUL entry is carried out onto the transformed 2d axial mesh.
5. In the time step computation the circumferential mesh-size will not be taken into account.
  6. Use option PHI2 with caution. Euler initialization is done using the mesh rotated by the angle PHI2. So after including the angle PHI2 or modifying its value the Euler initialization should be revised.
  7. It is assumed that one of the coordinate planes is an approximate symmetry plane of the Euler mesh. Although approximate symmetry is sufficient, the coordinate plane can always be made an exact symmetry plane by the use of PHI2. If for example the Euler mesh has angles 0 and 2.5, PHI2 has to be set to -1.25 to get exact symmetry.

**DYPARAM,AXREMAP**

2D Axial symmetric Euler archive remap - SOL 700

Allows import of a 2D axial symmetric Euler archive into a 3D simulation.

Format:

| 1       | 2       | 3  | 4  | 5  | 6  | 7  | 8  | 9     | 10 |
|---------|---------|----|----|----|----|----|----|-------|----|
| DYPARAM | AXREMAP | x0 | y0 | z0 | xn | yn | zn | range |    |

Example:

|         |         |     |     |     |    |    |    |  |  |
|---------|---------|-----|-----|-----|----|----|----|--|--|
| DYPARAM | AXREMAP | 0.0 | 0.5 | 0.5 | 1. | 0. | 0. |  |  |
|---------|---------|-----|-----|-----|----|----|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| x0,y0,z0   | x,y,z-coordinate of the point at which the 2D axial symmetric mesh is remapped. (Real; Default = 0.0)   |
| xn,yn,zn   | Unit vector that specifies the direction of the axial axis of the 2D axial symmetric mesh as viewed in the 3D mesh. (Real; Default = (1.0., 0.0, 0.0))        |
| range      | Only material whose distance from the axial axis is smaller than "range" will be initialized with the 2D axi-symmetric Euler archive. (Real, Default = 1e+20) |

Remarks:

1. Since 2D axial symmetric simulations run much faster than 3D simulation it can save much cpu time to do the first part of the simulation with a 2D axial symmetric mesh. Afterwards the 2D-axial symmetric Euler archive is imported into the 3D simulation. By default the 2D axial symmetric archive will not be expanded in 3D. To enable this expansion DYPARAM, AXREMAP has to be used. It is useful for blast wave simulations. The 2D axial symmetric simulation has to be terminated before the blast wave approaches any 3D structure.
2. This import of Euler archives is done by means of the eid option in the pth file.
3. To generate axial symmetric meshes DYPARAM,AXIALSYM can be used.

## DYPARAM,BULKL

Linear Bulk Viscosity Coefficient- SOL700

Defines the default value of the linear bulk viscosity coefficient.

**Format:**

DYPARAM, BULKL, VALUE

**Example:**

DYPARAM, BULKL, 0.1

| Descriptor | Meaning   |
|------------|---|
| value      | Value of the linear coefficient in the bulk viscosity equation. (Real $\geq 0.0$ ; default=0.0) |

**Remarks:**

1. The default value works well for the majority of problems.
2. The value defined on this entry is used as the default whenever BULKL is blank on the MATBV material entry.
3. When BULKL is specified on a material definition entry, the default value is overridden for that specific material.

**DYPARAM,BULKQ**

## Quadratic Bulk Viscosity Coefficient - SOL700

Defines the default value of the quadratic bulk viscosity coefficient.

**Format:**

DYPARAM, BULKQ, VALUE

**Example:**

DYPARAM, BULKQ, 1.6

| Descriptor | Meaning  |
|------------|--|
| value      | Value of the quadratic coefficient in the bulk viscosity equation. (Real $\geq 0.0$ ; default=0.0) |

**Remarks:**

1. The default value works well in the majority of situations.
2. The value defined on this entry is used as the default whenever BULKQ is blank on the MATBV material entry.
3. When BULKQ is specified on a material definition entry, the default value is overridden for that specific material.

## DYPARAM,BULKTYP

## Bulk Viscosity Type - SOL700

Defines the default type of bulk viscosity.

**Format:**

DYPARAM, BULKTYP, option

**Example:**

DYPARAM, BULKTYP, DYNA

| Describer | Meaning  |
|-----------|--|
| Option    | bulk viscosity type. (Character; Default=DYNA) |
| DYNA      | Standard DYNA3D model                          |
| DYTRAN    | Enhanced DYNA model                            |

**Remarks:**

1832

## DYPARAM,CFULLRIG

Converts 123456 Constraints to FULLRIG on RBE2 - SOL700

### DYPARAM,CFULLRIG

Converts 123456 Constraints to FULLRIG on RBE2 - SOL700

Converts all 123456 constraints to the FULLRIG option on all entries. Used in SOL 700 only.

**Format:**

DYPARAM, CFULLRIG, value

**Example:**

DYPARAM, CFULLRIG, NO

| Descriptor | Meaning  |
|------------|--|
| value      | Activation flag. (Character; Default=YES)        |
| YES        | 123456 constraints are converted to FULLRIG.     |
| NO         | 123456 constraints are not converted to FULLRIG. |

**DYPARAM,CLUFLIM****Limiter of Volume Stain Rate for Clumps - SOL700**

In some cases, airbag runs become instable. Often, this is caused by a much too large volume strain rate in a clump that consists of too many elements. These clumps typically have a small average volume uncovered fraction. The large volume strain rate causes a huge compression work and this blows up the specific internal energy. When this happens it is clearly visible in the OUT file and in the results. This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable. Used in SOL 700 only.

**Format:**

DYPARAM, CLUFLIM, value

**Example:**

DYPARAM, CLUFLIM, 0.22

| Descriptor | Meaning  |
|------------|--|
| value      | The volume strain rate in a clump will be reduced when the average Uncovered Fraction of elements in a CLump falls below CLUFLIM. CLUFLIM has to be smaller than FBLEND. The default value of FBLEND is 0.66 giving a value of 0.22 for CLUFLIM. For more details on FBLEND refer to PARAM,FBLEND. (0<Real<FLBEND; default=FBLEND/3) |

**Remarks:**

1. The Volume strain rate  $\frac{DIV}{\Delta t}$  in clumps will be limited by

$$\left( \frac{DIV}{\Delta t} \right)_{Lim} = MIN\left( 1, \frac{U}{CLUFLIM} \right) \left( \frac{DIV}{\Delta t} \right)$$

Here, U is the average uncovered fraction of elements in the clump as given by

$$U = \frac{\sum_{eleclump} Uncf_{el} \cdot Vol_{el}}{\sum_{eleclump}}$$

Here Uncf and Vol denote the uncovered fraction and volume of an element inside the clump. Therefore, only when the average uncover fraction falls below CLUFLIM, the volume strain rate is limited.

**DYPARAM,CLUMPENR**

Switch for Kinetic Energy Calculation Scheme of Blended Clumps - SOL700

Sets the definition of the kinetic energy calculation method for Eulerian blended clumps. Used in SOL 700 only.

**Format:**

DYPARAM, CLUMPENR, value

**Example:**

DYPARAM, CLUMPENR, SUM

| Descriptor | Meaning  |
|------------|--|
| value      | Flag for CLUMPENR.(Character; default=AVERAGE)   |
| AVERAGE    | The kinetic energy of a Eulerian blended clump is calculated from the average velocity of the clump. The average velocity of the blended clump is computed as the sum of the momentum of each member of the clump divided by the total clump mass. |
| SUM        | The kinetic energy of a Eulerian blended clump is calculated as the sum of the kinetic energy of each member of the clump.   |

**DYPARAM,CONN2OUT**

## CONN2 Summary Output - SOL700

Determines if a summary of concentrated masses and their energy and momentum is written to the output file. Used in SOL 700 only.

**Format:**

DYPARAM,CONN2OUT,value

**Example:**

DYPARAM,CONN2OUT,NO

| Descriptor | Meaning  |
|------------|--|
| value      | Flag for COMN2OUT. (Character; default=NO)   |
| NO         | No information about concentrated masses is written to the cycle and material summaries on the output file.                  |
| YES        | A complete summary of concentrated masses including the associated mass, momentum, and energy is written to the output file. |

**Remarks:**

1. When DYPARAM,CONN2OUT is set to NO, there is no summary of the concentrated mass. This means that the mass, momentum, and energy of the concentrated masses, is not added to the material and cycle summaries. Setting DYPARAM,CONN2OUT,NO saves memory and CPU time.

## DYPARAM,CONTACT

Sets Defaults for CONTACT - SOL700

Defines certain defaults for the contact definitions. Used in SOL 700 only.

**Format:**

DYPARAM,CONTACT,option,value1,value2,value3,value4,value5,value6

**Example:**

DYPARAM,CONTACT,VERSION,V4

| Descriptor | Meaning  |
|------------|--|
| VERSION    | Defined the default version of contact. Only support value1. (Character; required)   |
| V4         | General Contact algorithm of SOL700  |
| BELT       | Suited for modeling contact between a belt element and a rigid structure.<br>Master slave contact only.<br>The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the slave nodes exactly on top of the master face.<br>The slave node does not slide relative to the master face when the friction coefficient (FS) is set to 1E20.  |
| BELT1      | Identical to BELT algorithm, except that the slave nodes are initially repositioned on top of the closest master face.   |
| DRAWBEAD   | Suited for modeling a drawbead.  |
| THICK      | Defines the default value for THICK. Only support vaule1. (Real>0.0; required.)  |
| GAP        | Defines the default value for GAP. Only support vaule1. (Real>0.0; required.)  |
| LIMITS     | Definition of a three dimensional contact region where contact in the analysis model takes place. Significant CPU time savings can be achieved when used in adaptive contact. 6 values are used to define area.<br><br>The format of this option is<br>DYPARAM,CONTACT,LIMITS,xmin,xmax,ymin,ymax,zmin,zmax.<br>(Real; default=1.0E20 for xmax, ymax and zmax and -1.0E20 for xmin, ymin and zmin)<br>XMIN(value1) Lower limit in x-direction where main contact occurs<br>XMAX(value2) Upper limit in x-direction where main contact occurs<br>YMIN(value3) Lower limit in y-direction where main contact occurs<br>YMAX(value4) Upper limit in y-direction where main contact occurs<br>ZMIN(value5) Lower limit in z-direction where main contact occurs<br>ZMAX(value6) Upper limit in z-direction where main contact occurs |
| DAMPING    | Defines the default for usage of damping. Only support value1. (Character; required)   |
| YES        | Use damping in CONTACT   |

| Descriptor | Meaning           |   |
|------------|-------------------|---|
|            | NO                | Not use damping in CONTACT  |
| COPOR      |                   | Activates contact based porosity. Only support value1. (Character; default=NO)  |
|            | YES               | Use porosity in CONTACT   |
|            | NO                | Not use porosity in CONTACT   |
| DAMPFOR    |                   | Defines whether the noncontact forces acting on the grid points need to be taken into account in the contact damping. This option is only used if DAMPING is set to YES. This option prevents large penetrations that might occur when the forces acting on the grid points tend to push them into the contact surface. This happens, for example, in airbag analyses, where a large pressure exists inside the bag. Only support value1. (Character; required) |
|            | YES               | Damping is considered in noncontact forces  |
|            | NO                | Damping is not considered in noncontact forces  |
| DYNA       |                   | The following parameters of the contact definition get the default values consistent with Dyna. (Character ; required)  |
|            | THICK             | 1.0   |
|            | THICKOF           | 0.0   |
| INFO       |                   | Information on the contact state of grid points G1,G2,... is printed to ASCII files, named CNT. This information can be useful in debugging models with contacts. Supports many values as you want. (Integer > 0; required.)  |
| EVIEW      |                   | Defines the default value of the view angle of Edge to Edge contacts. The value of the angle must be in degrees. Only support value1. (Real > 0.0; required)  |
| FORCE      |                   | Controls the contact forces on the grid points. Supports value1, value2, value3. See Remark 1. (value1: Integer > 0; default=10, value2: Real > 0; default=10.0; value3: Character; default=ZERO)   |
|            | NMCYC<br>(value1) | Frequency check of contact force; applied on each grid point. (Integer > 0; default = 100).   |
|            | SCALE<br>(value2) | Scale factor for maximum allowable contact force.<br>$F_{max} = SCALE \cdot F_{last\_check}$ (Integer > 0; default = 10)  |
|            | TYPE<br>(value3)  | Contact force limitation. (Character; default=ZERO)   |
|            | ZERO              | 0.0   |
|            | FMAXF             | Update contact limit forces using new contact forces (where non allowable forces are not taken into account)  |

**Remarks:**

1. The DYPARAM, FORCE check takes up some CPU time and, therefore, do not make this value too small. Furthermore, when the check is performed at each cycle, the force will be too limited and the bag will not unfold. Recommended values are between 5 and 200. The same problems can occur for SCALE (value2). In case this value is too small, the bag will not unfold either. The minimum value for air bags that should be used is about 5. The maximum is about 20. When this value is too big a difference will not be noticed. TYPE (value3)=ZERO is a bigger restriction. In some cases, TYPE=FMAXF might yield better results.

**DYPARAM,COSUBMXT**

Time-dependent Subcycle Limit in Euler/Lagrange Coupling - SOL700

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table. Used in SOL 700 only.

**Format:**

DYPARAM,COSUBMXT,value

**Example:**

DYPARAM,COSUBMAT,10

| Descriptor | Meaning   |
|------------|---|
| value      | TABLED1 ID that specifies for each time the maximum number of time steps between updating the coupling surface geometry in the coupling calculations. (Integer > 0; required) |

**Remarks:**

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values on the table should be used.

**DYPARAM,COUFRIC**

Coupling Surface Friction for Nonmetallic Eulerian Solids - SOL700

Defines the Coulomb friction scheme. Please check DYPARAM,COHESION too. Used in SOL 700 only.

**Format:**

DYPARAM, COUFRIC, value

**Example:**

DYPARAM, COUFRIC, NO-METAL

| Descriptor | Meaning  |
|------------|--|
| value      | Flag for COUFRIC usage. (Character; default=METAL)                                       |
| METAL      | A tensile condition will result in zero load on the structure part on cohesive coupling. |
| NO-METAL   | a tensile load was applied in tensile condition on cohesive coupling.                    |

**Remarks:**

1. Only used when Coulomb friction coefficients have been specified for a COUPLE entry.

**DYPARAM,COHESION**

Cohesion for Coulomb Friction - SOL 700

Allows friction and sticking during tensile conditions at the coupling surface.

**Format:**

DYPARAM,COHESION,MAXSTRS,FRIC,REFVEL

**Example:**

DYPARAM,COHESION,8.0e+10,8.0e+5,20.0

| Descriptor | Meaning   |
|------------|---|
| MAXSTRS    | Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real > 0) |
| FRIC       | Friction stress under tensile conditions. (Real > 0)  |
| REFVEL     | Reference value for velocity. (Real > 0)  |

**Remarks:**

1. Only used when coulomb friction coefficients have been specified for a COUPLE entry.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals:

$$\text{FRIC} \cdot \min\left(1, \frac{V_{\text{rel,tangential}}}{\text{REFVEL}}\right)$$

This is a viscous-like friction law.

3. This shear force opposes the relative tangential movement along the coupling surface.

**DYPARAM, DUMSEGS**

Creating dummy segments for the auto coupling approach - SOL700

Specifies how holes in the coupling surface are meshed. It is only used for the auto coupling approach. This approach is activated by DYPARAM, AUTOCOUP.

**Format and Example:**

DYPARAM,DUMSEGS,INTERNAL

| Descriptor | Meaning  |
|------------|--|
| INTERNAL   | Meshes internal holes with a triangular dummy segment mesh [YES,NO] Default NO |

**Remarks:**

- With the auto coupling method, holes are partially or completely meshed with dummy segments. With option NO the standard method of creating dummy segment is used. For details see chapter 8 of the SOL700 User's Guide. In this standard method holes are partially meshed with dummy segments. For an example refer to Figure 20 of Chapter 8. For holes that rotate, have arbitrary form or show large deformations, the standard method of creating dummy segments can cause the simulation to terminate with messages like

```
%E-P4307807-P4_SUBELM_MESH_CURVED_HOLES, , ,(13), DUMMY SURFACE  
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT  
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION  
PROBLEM.
```

```
%E-P4309907-V4_SUBELEM_CONNECT_EXTENDED_GPS, , DUMMY SURFACE  
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT  
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION  
PROBLEM.
```

In that case PARAM,DUMSEGS,ON can maintain a good dummy surface. The quality of the dummy surface can be checked by requesting a coupling surface archive. This is done by using the CPLSURFS command and by putting option DUMMY of DYPARAM, AUTOCOUP to ON. For all holes that are not internal holes, the standard method is used.

**DYPARAM,ELDLTH**

- SOL700

Print initial time step sizes for elements in the first cycle in SOL 700.

**Format:**

DYPARAM,ELDLTH,<value>

**Example:**

DYPARAM,ELDLTH,1

| Describer | Meaning   |
|-----------|---|
| value     | Number of elements to be output. (Integer)                            |
| 0         | First 100 elements with the smallest time step sizes are printed.     |
| 1         | The governing time step sized for all elements are printed. (Default) |

**DYPARAM,EULERCB**

Euler Domain Decomposition Control - SOL700

Divides a Euler domain into several cubes. For SOL 700 only

**Format:**

DYPARAM,EULERCB,NELCUBE,NBX,NBY,NBZ

**Example:**

DYPARAM,EULERCB,2000,2,2,2

| Descriptor | Meaning  |
|------------|--|
| NELCUBE    | The number of elements per cube. (Integer; Default = 0)                      |
| NBX        | The number of cubes in the x-direction. See Remark 2. (Integer; Default = 0) |
| NBY        | The number of cubes in the y-direction. (Integer; Default = NBX)             |
| NBZ        | The number of cubes in the z-direction. (Integer; Default = NBX)             |

**Remarks:**

1. By setting NELCUBE equal to 2000 optimal use is made of memory caching during Euler computation. This may give a speedup of 1.5. When using adaptive meshing with dmp additional Euler cubes are created during the simulation. To keep the total number of cubes that are created within bounds the initial number of cubes should be limited to 100.
2. Defining NBX overrules the definition of NELCUBE.
3. Only supports Euler domains created by MESH,BOX. Limitations are:
  - No PEULERx/CHEXA's
  - No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.

**DYPARAM,EULERPR****Euler Cube Partition Control - SOL700**

Divides a Euler domain into several cubes. For SOL 700 only

**Format:**

DYPARAM,EULERPR,PROCDIR,NPX,NPY,NPZ

**Example:**

DYPARAM,EULERPR,USER,2,2,2

| Descriptor | Meaning   |
|------------|---|
| PROCDIR    | This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Character; Default = X) |
| X          | Partition in global X direction first.  |
| Y          | Partition in global Y direction first.  |
| Z          | Partition in global Z direction first.  |
| USER       | Define user defined partitioning  |
| SIMPLE     | Partition Euler cubes in a simple pattern.  |
| ORB        | Partitions using orthogonal recursive bisection   |
| NPX        | The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer; Default = 0)   |
| NPY        | The number of cubes in the y-direction. (Integer; Default = NPX)  |
| NPZ        | The number of cubes in the z-direction. (Integer; Default = NPX)  |

**Remarks:**

1. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.
2. When option PROCDIR=USER, the values for NBX, NBY and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option, NPX\*NPY\*NPZ must be equal to the number of processors used in the cluster.
3. For option PROCDIR=SIMPLE, the values NBX, NBY and NBZ on DYPARAM,EULERCB must be such that NBX\*NBY\*NBZ is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4, NBX\*NBY\*NBZ must be equal to either 4 or 8 or 12, etc. Otherwise the calculation will terminate prematurely with an error message.
4. Only supports Euler domains created by MESH,BOX. Limitations are:
  - No MESH,ADAPT and no PEULERx/CHEXA's

- No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.
5. With the orthogonal recursive bisection method the mesh is repeatedly cut in half. Each cut is along a coordinate direction and the direction of the cut is chosen to minimize the communication cost. To illustrate this method consider for example a mesh with 200,150 and 50 elements in respectively the x,y and z-direction. When split in x-direction the number of Euler faces at the split will be  $150*50 = 7500$ . When splitting across the y-direction it will be  $200*50=10000$  and for the z-direction it will be  $200*150=30000$ . Since the x-split has the smallest number of faces at the split, the x-split has the smallest communication cost. Therefore the ORB scheme will select the x-split. At the x-split there are two mesh parts. Both have  $100*150*50$  elements. Now a split in y-direction will give the minimal communication costs. This process of bisecting is continued until the number of sub meshes equals the number of CPU's.

**DYPARAM,EULTRAN**

Switch for Euler Transport Scheme of the Multi-Material Solver and the Single Material Strength Solver - SOL700

Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver. Used in SOL 700 only.

**Format:**

DYPARAM,EULTRAN,option1,option2

**Example:**

DYPARAM,EULTRAN,AVERAGE,FAIL

| Descriptor | Meaning |  |
|------------|---------|--|
| Option1    | IMPULSE | The face velocity is impulse weighted. (Character; Default = IMPULSE)      |
|            | AVERAGE | The face velocity is a simple average.                                     |
| Option2    | NOFAIL  | Failure is transported. See Remark 5. and 6. (Character; Default = NOFAIL) |
|            | FAIL    | Failure is transported. See Remark 5. and 6.                               |

**Remarks:**

1. The default value of IMPULSE is sufficient for most Euler problems. Especially problems where the reference density of the different materials varies widely (e.g., orders of magnitude) are required to use the default option.
2. In case the IMPULSE option (default) is used, the Euler transport scheme computes that the face velocity uses an impulse weighted average of the material velocity in the left and the right element adjacent to the face.
3. In case the AVERAGE option is used, the Euler transport scheme computes the face velocity as one-half times the sum of the material velocity in the left and the right element adjacent to the face.
4. Does not apply to the single material hydrodynamic solver and the Roe solver.
5. The option FAIL requires a failure model for at least one Eulerian material. In case of the default NOFAIL then failed Euler material can support shear stress again as soon as new material enters the Euler element. Thus the information that part of the material inside the Euler element has failed is lost. The option FAIL activates transport of fail fraction and thereby keeps track of material that has failed. In this way only the failed part of the element can no longer supports shear stresses. In more detail, the yield stress in the material is scaled by (1-failfrac), where failfrac denotes the fail fraction of the material. The fail fraction of the first material in an element can be retrieved from Euler archive or time-history results files in the variable DAMAGE. The value of fail fraction DAMAGE is between zero and one.
6. Option FAIL cannot be combined with the Johnson-Cook failure model (FAILJC).

**DYPARAM,EUSUBMAX**

- SOL700

Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped. Used in SOL 700 only.

If coupling surface computations are more expensive than Euler computations than use of PARAM,COSUBMAX should be considered first. In that case the optimal setting of EUSUBMAX is LINKCS=BOTH and DFVUMAX is blank.

As with PARAM,COSUBMAX use of EUSUBMAX can lead to loss of accuracy for certain simulations. It is recommended to validate the use of EUSUBMAX by comparing the difference in results between using EUSUBMAX and not using EUSUBMAX for some typical target simulations.

**Format**

DYPARAM,EUSUBMAX,NSUBMAX,DFVUMAX,LINKCS

**Example:**

DYPARAM,EUSUBMAX,5

| Descriptor | Meaning  |
|------------|--|
| NSUBMAX    | NSUBMAX, The maximum number of time steps between updating Euler variables.<br>(Integer $\geq 0$ ; Default = 0)  |
| DFVUMAX    | Maximum allowed increase in uncovered volume fraction between to subsequent Eulerian computations. (Real $> 0$ ; Required when COSUBMAX is present. Leave blank when COSUBMAX is not present.)   |
| LINKCS     | Specifies interactions between Euler subcycling and coupling surface subcycling.<br>EUSUBMAX can be used with COSUBMAX. LINKCS specifies how the two subcycling processes influence each other. (Character; Default = BOTH)  |
| BOTH       | The Euler computations will not be skipped when the coupling surface computations has been done. On the other hand if an Euler computation is to be done, also a coupling surface computation is done. The number of time that the Euler computation I skipped equals the number of times that the coupling surface computation has been done. |
| COUPLE     | If an Euler computation is to be done, also a coupling surface computation is done.  |
| EULER      | If a coupling surface computation is to be done, also a Euler computation is done.   |
| NONE       | The Euler subcycling and coupling surface subcycling are independent.<br>DFVUMAX has to be defined.  |

Remarks:

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of DFVUMAX, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. When the Euler time step is considerably larger than the Lagrange time step it can be worthwhile to skip the Euler computation for several cycles. This can reduce computational costs considerable.
4. Skipping the Euler computation for several cycles, leads to postponed time steps. When the Euler computations are done these have to be taken into account by fluxing with an accumulated time step. Skipping the Euler computation is stopped as soon as the accumulated time step will become larger than the stable Euler time step. The total number of skipped cycles is limited by NSUBMAX. NSUBMAX is required input.
5. In skipping the Euler computation also the movement of the coupling surface has to be monitored. To estimate this movement the change in uncovered fraction of the elements is used. If the change in uncovered volume fraction from one cycle to the other is larger than DFVUMAX than the Euler computation is not skipped that cycle. DFVUMAX has to be left blank if coupling surface subcycling is used. Coupling surface subcycling already monitors the movement of the coupling surface. Choosing too large values for DFVUMAX can make results inaccurate.
6. EUSUBMAX is only supported by the multi-material Euler solver and the standard single material Euler solver. It is not supported by the Roe solver.

**DYPARAM,EUSUBCYC**

## Growth or Subcycling Interval in Euler Computations - SOL700

Controls the growth of the subcycling interval in Euler computations. Used in SOL 700 only.

**Format:**

DYPARAM,EUSUBCYC,value

**Example:**

DYPARAM,EUSUBCYC,100

| Descriptor | Meaning   |
|------------|---|
| value      | Maximum growth of the subcycling interval. (Integer > 0; default=1) |

**Remarks:**

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If Dytran estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

**DYPARAM,FAILDT**

## Element Time-step Based Failure Model - SOL700

Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit. Used in SOL 700 only

**Format:**

DYPARAM, FAILDT, value

**Example:**

DYPARAM, FAILDT, 1.0E-3

| Descriptor | Meaning  |
|------------|--|
| value      | Minimum time-step. (Real > 0.0; default=1.0E-20) |

**Remarks:**

1. This failure model is valid for all Lagrangian solid (CHEXA) and shell (CQUAD4) elements.
2. All elements for which the time step falls below the specified value are removed from the computation.
3. Although it is not usually necessary to limit the element time-step for Lagrangian elements, there are occasions where specifying a minimum time-step can be advantageous for computational performance, for example, when adaptive contact is used.
4. Note that this parameter should be used with care as you may influence the results of the analysis when you set the time-step criterion to a too high value. You then run the risk that elements are removed from the analysis while they may still be relevant.

## DYPARAM,FAILOUT

Failed Element Output Parameter - SOL700

Defines whether failed elements are written to the output file (ARCHIVES). Used in SOL700 only.

**Format:**

DYPARAM, FAILOUT, value

**Example:**

DYPARAM, FAILOUT, NO

| Descriptor | Meaning   |
|------------|---|
| value      | Flag for FAILOUT. (Character; default=YES)            |
| NO         | Failed elements are not written to the archive files. |
| YES        | Failed elements are written to the archive files.     |

**Remarks:**

1. When the NO option is chosen, the archives are written out as one file per requested time step regardless of the number set in the SAVE command for the archive files that appear in the Case Control Section. Please check \$S700.

**DYPARAM, FLOWMETH****Flow-Method Between Two Euler Domains Across Open Areas in Coupling Surfaces - SOL700**

Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces. Used in SOL 700 only.

**Format:**

DYPARAM, FLOWMETH, value

**Example:**

DYPARAM, FLOWMETH, POLPACK

| Descriptor | Meaning   |
|------------|---|
| value      | Flag for FLOWMETH. (Character; default=NO)  |
| POLPACK    | The facets in the coupling surfaces that represent an open area are subdivided into smaller facets, with each connecting exactly to one Euler element in the first Euler domain and to exactly one Euler element in the second Euler domain. Material flow takes place across these smaller, subdivided facets (POLPACKs). This is the most accurate method.  |
| FACET      | The facets in the coupling surfaces that represent an open area are not subdivided. Material flow takes place across the original facets. If these facets are too large in relation with the Euler elements, the method becomes inaccurate. Material flow across one facet can involve several Euler elements on both sides of the hole and averaging occurs. |

**Remarks:**

1. This parameter applies to simulations where:
  - Two coupling surfaces share a common set of facets.
  - Each coupling surface has own Euler domain.
  - Material flows from one Euler domain into the other through the open area represented by the common set of facets.

Flow only occurs if:

- The common facets are defined as 'open', using PORFCPL.
- The common facets open up due to failure of a shell structure, using COUP1INT.

Examples simulations are:

- Holes between air bag compartments.
- Holes between containers filled with gas or liquid.
- Open area between the top of a fuel-tank baffle and the fuel-tank skin.

- Open area in-between wide straps inside an air bag.
  - Failure of walls in between aircraft wing compartments.
  - Failure of tank armor by a blast wave.
  - Etc.
2. The following table summarizes what input cards support the simulation of material flow between two Euler domains across open areas in coupling surfaces:

| Euler Solver                            | Material flow through a coupling surface | FLOW-METHOD=POLPACK | FLOW-METHOD=FACET | PORFLCPL (velocity based) | PORFLCPL (Pressure based) | COUP1INT/ COUP1FL (Failure of shell elements creates the opening) |
|---|--|---------------------|-------------------|---------------------------|---------------------------|---|
| HYDRO                                   | YES                                      | YES                 | NO                | YES                       | YES                       | Only for flow method = polpack                                    |
| HYDRO-Roe solver-1 <sup>st</sup> Order  | YES                                      | YES                 | YES               | YES                       | YES                       | YES   |
| HYDRO-Roe solver- 2 <sup>nd</sup> Order | YES                                      | YES                 | YES               | YES                       | YES                       | YES   |
| MMHYDRO                                 | YES                                      | YES                 | NO                | YES                       | NO                        | Only for flow method = polpack                                    |
| STRENGTH                                | NO                                       | -                   | -                 | -                         | -                         | -   |
| MMSTREN                                 | YES                                      | YES                 | NO                | YES                       | NO                        | Only for flow method = polpack                                    |

3. The Euler domains are shown below with the support types for each:

|                            | FLOW-METHOD=POLACK | FLOW METHOD = FACET |
|----------------------------|--------------------|---------------------|
| MESH → TYPE= ADAPT         | YES                | No                  |
| MESH → TYPE= BOX           | YES                | YES                 |
| Modeling of CHEXA elements | No                 | YES                 |

4. A Euler domain is associated with a coupling surface by specifying the MESHID or SET1ID on the COUPLE option.
5. FLOWMETH = POLPACK has the following limitations:

The entries NSTGP and NSTEL on all MESH entries should be left blank. It is not allowed to specify for any MESH entry the starting element number or starting grid point number.

- There are restrictions on output requests.
- Flow faces and wallets are not supported while FLOWDEF is supported.
- Viscosity is not supported

A case where these limitations require the use of FLOWMETH = FACET is when the Euler elements are generated in Patran, not using the MESH option, and one or more of the following options is used:

- FLOW boundaries are defined on some Euler faces.
- WALLET boundaries are defined on some Euler faces.
- Viscosity is defined.

## DYPARAM,GEOCHECK

Define Geometry Consistency Check - SOL700

This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed. Used in SOL 700 only.

**Format:**

DYPARAM, GEOCHECK, value

**Example:**

DYPARAM, GEOCHECK, ON

| Descriptor | Meaning  |
|------------|--|
| value      | Flag for activating GEOCHECK. (Character; default=OFF) |
| ON         | Geometry consistency check is performed.               |
| OFF        | No geometry consistency check is performed.            |

**Remarks:**

1. The defined geometry is checked for consistent connectivity of the hexagonal elements. If an inconsistency is detected, the connectivity is corrected. CFACE entries with references to elements that have been corrected are corrected as well.
2. If a hexagonal mesh is generated with other commercial preprocessors, this parameter can correct the connectivity of the hexagonal elements in case problems are encountered with face generation or volume computation.

## DYPARAM,FASTCOUP

## Fast Coupling Algorithm - SOL700

Defines the fast coupling algorithm. Used in SOL 700 only.

**Format:**

DYPARAM,FASTCOUP,option1,option2,option3,option4,option5,option6,option7

**Example:**

DYPARAM,FASTCOUP,INPLANE,FAIL

| Descriptor | Meaning   |
|------------|---|
| Option1    | NO            Fast coupling interaction is turned off. (Character; Default = INPLANE)   |
|            | INPLANE      Small offset for inplane coupling surface segments.  |
|            | NOOFFSET     No offset for inplane coupling surface segments  |
| Option2    | NOFAIL        No failure of the coupling surface. (Character; Default = NOFAIL)   |
|            | FAIL           Failure of the coupling surface will be taken into account.  |
| Option3    | Euler check algorithm. (Character; Default = NOCHKEUL)  |
|            | CHKEUL        Checking whether all segments of the coupling surface are fully inside Euler elements.  |
|            | NOCHKEU      No checking<br>L   |
| Option4    | Euler update algorithm. (Character; Default = ALL)  |
|            | NEARONLY      Do the full coupling surface only in the first cycle. In subsequent cycles only update elements and faces that are near the coupling surface. |
|            | ALL            Do the full coupling surface computation each cycle.   |
| Option5    | Controls coupling surface computations when there are multiple cubes defined with DYPARAM,EULERCB. (Character; Default = NONE)                              |
|            | NONE           All coupling surface computations are done for each cube.  |
|            | FACES          The face cover fraction computation is done in one go for all cubes. The other coupling computations are still done per cube.                |
|            | POLPK          The polpack computation efficiently handles multiple cubes. The other coupling computations are still done per cube.                         |
| ALL        | All coupling surface computations efficiently handle cubes.   |

| Descriptor | Meaning   |
|------------|---|
| Option6    | In the coupling surface computations it is determined which structural segments intersect the Euler mesh. This is done by looping across all structural segments. For dmp each cpu has only part of the Euler mesh. Therefore for many segments it is known beforehand that they will not intersect the Euler mesh of the cpu and can be skipped at an early stage of the computation. Option 6 activates this skipping. (Character; Default = ON)  |
| OFF        | For dmp each CPU goes over all segments.  |
| ON         | Each CPU skips all segments that do not intersect the Euler mesh of the CPU.  |
| Option7    | Each coupling surface segment has to be intersected with the Euler elements and Euler faces. For large Euler meshes a substantial amount of checking has to be done to find the intersecting elements and faces. By dividing the Euler mesh in search boxes the costs of this checking is kept to a minimum. Especially for large Euler meshes combined with large coupling surface this checking can become expensive and then the use of search boxes can significantly reduce the costs. For Euler meshes of less than 100000 elements the use of search boxes have little effect. (Character; Default = ON) |
| OFF        | Search boxes are not used.  |
| ON         | Search boxes are used.  |

#### Remarks:

1. The fast coupling algorithm is always turned on by default. In order to use general coupling, option1=NO should be used.
2. When option1 is set to INPLANE or when option 1 is blank, a small offset is given to coupling surface segments that are on top of a face of an Eulerian element. This is done because coupling surfaces segments on Eulerfaces make the Euler element volume computation invalid. Also boundary conditions on these segments are not correctly imposed. The net effect of these problems is unpredictable. The problem can either run correctly, or remain stable but give false results or become instable.
3. Option2 can only be used in combination with either PARAM,LIMITER,ROE or MMHYDRO or MMSTREN. The coupling surface must consist of CQUADS and/or CTRIAs and a failure model for the material of the surface must be defined.
4. This parameter can only be used when the Eulerian mesh is aligned with the basic coordinate system axes.

**DYPARAM,HYDROBOD****Hydro Body Force - SOL700**

Defines a body force for single hydro material in Euler. Used in MSC Nasran SOL 700 only.

**Format:**

DYPARAM,HYDROBOD,XACC,YACC,ZACC

**Example:**

DYPARAM,HYDROBOD,-300.,0.,150.

| Descriptor | Meaning                               |
|------------|---------------------------------------|
| XACC       | X-acceleration (Real; Default = 0.0)  |
| YACC       | Y-acceleration. (Real; Default = 0.0) |
| ZACC       | Z-acceleration. (Real; Default = 0.0) |

**Remark:**

1. This parameter defines a constant body force load in Euler for single hydro material only.

**DYPARAM,HGCMEM**Shell Membrane Hourglass Damping Coefficient  
Parameters - SOL700

Defines the default membrane damping coefficient for shell elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGCMEM, value

**Example:**

DYPARAM, HGCMEM,0.07

| Descriptor | Meaning   |
|------------|---|
| value      | Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.) |

**Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

**DYPARAM,HGCOEFF****Hourglass Damping Coefficient - SOL700**

Defines the global default hourglass damping coefficient. Used in SOL 700 only.

**Format:**

DYPARAM, HGCOEFF, value

**Example:**

DYPARAM, HGCOEFF, 0.14

| Descriptor | Meaning   |
|------------|---|
| value      | Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.) |

**Remarks:**

1. The default applies to all types of hourglass suppression methods and should be used unless there is good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficients are not explicitly defined on HGSUPPR entries or on a HGCMEM HGCWRP, HGCTWS, or HGCSOL entry.
3. If this entry is omitted, the default value of the hourglass damping coefficients is either equal to the default value of 0.1 or is equal to the value specified on a HGCMEM, HGCTWS, HGCWRP, or HGCSOL DYPARAM entry.
4. The value of the coefficients can be explicitly defined for each property by using an HGSUPPR entry.

**DYPARAM,HGCSOL**

## Solid Hourglass Damping Coefficient - SOL700

Define the default damping coefficient for solid elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGCSOL, value

**Example:**

DYPARAM, HGCSOL, 0.11

| Descriptor | Meaning   |
|------------|---|
| value      | Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.) |

**Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for solid elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

## DYPARAM,HGCTWS

## Shell Twisting Hourglass Damping Coefficient - SOL700

Defines the default twisting damping coefficient for shell elements. Used in SOL 700 only.

Format:

DYPARAM, HGCTWS, value

Example:

DYPARAM, HGCTWS, 0.02

| Descriptor | Meaning  |
|------------|--|
| value      | Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark <a href="#">3</a> .) |

Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

**DYPARAM,HGCWRP**

## Shell Warping Hourglass Damping Coefficient - SOL700

Defines the default warping damping coefficient for shell elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGCWRP, value

**Example:**

DYPARAM, HGCWRP, 0.0

| Descriptor | Meaning   |
|------------|---|
| value      | Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.) |

**Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

**DYPARAM,HGSHELL****Shell Hourglass Suppression Method - SOL700**

Defines the default hourglass suppression method for shell elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGSHELL, value

**Example:**

DYPARAM, HGSHELL, DYNA

| Descriptor | Meaning   |
|------------|---|
| value      | Type of HGSHELL. (Character; default=See remark 2.) |
| FBV        | Flanagan-Belytschko viscous hourglass damping       |
| DYNA       | Viscous hourglass damping                           |

**Remarks:**

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for shell properties are left blank.
2. If this entry is omitted, the default suppression method used for shell elements is either FBV or the default method defined on the DYPARAM,HGTYPE entry.

**DYPARAM,HGSOLID**

## Solid Hourglass Suppression Method - SOL700

Defines the default hourglass suppression method for solid elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGSOLID, value

**Example:**

DYPARAM, HGSOLID, FBS

| Descriptor | Meaning   |
|------------|---|
| value      | Type of HGSOLID. (Character; default=See remark 2.) |
| FBS        | Flanagan-Belytschko stiffness hourglass damping     |
| DYNA       | Viscous hourglass damping                           |

**Remarks:**

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for solid properties are left blank.
2. If this entry is omitted, the default suppression method used for solid elements is either FBS or the default method defined on the DYPARAM,HGTYPE entry.

**DYPARAM,HGTYPE**

Hourglass Suppression Method - SOL700

Defines the default type of hourglass suppression method. Used in SOL 700 only.

**Format:**

DYPARAM, HGTYPE, value

**Example:**

DYPARAM, HGTYPE, FBS

| Descriptor | Meaning  |
|------------|--|
| value      | Type of HGTYPE. (Character; default=See remark 2.) |
| FBS        | Flanagan-Belytschko stiffness hourglass damping.   |
| FBV        | Flanagan-Belytschko viscous hourglass damping.     |
| DYNA       | Viscous hourglass damping.                         |

**Remarks:**

1. The type of the hourglass suppression method defined on this entry is used as the default whenever the type fields in the HGSUPPR entries are left blank.
2. If this entry is omitted, the type can be defined on a DYPARAM,HGSHELL entry for shell elements, a DYPARAM,HGSOLID entry for solid elements, or on the HGSUPPR entries; otherwise the defaults apply. For shell elements the default is FBV; for solid elements, the default is FBS.

## DYPARAM,HICGRAV

Gravity Used by HIC Calculations - SOL700

Defines the value of the gravity to be used by the HIC calculations. Used in SOL 700 only.

**Format:**

DYPARAM, HICGRAV, value

**Example:**

DYPARAM, HICGRAV, 980.7

| Descriptor | Meaning   |
|------------|---|
| value      | Gravity used by HIC Calculations. (Real > 0.0; default=9.80665) |

**Remarks:**

1. The value set by this parameter will be used by all HIC output requests.
2. This parameter can only be set once in the input deck.

## DYPARAM,HVLFAIL

Failure at Hydrodynamic Volume Limit - SOL700

Defines element failure on the hydrodynamic volume limit. Used in SOL 700 only.

Format:

DYPARAM, HVLFAIL, value

Example:

DYPARAM, HVLFAIL, YES

| Descriptor | Meaning   |
|------------|---|
| value      | Flag of HVLFAIL. (Character; default=NO)        |
| YES        | Element failure on hydrodynamic volume limit    |
| NO         | No element failure on hydrodynamic volume limit |

Remarks:

1. Lagrangian elements (CHEXA) that have a material model with a failure model fail when the hydrodynamic volume limit is reached and the parameter is set to YES.  
The elements can fail only when the following items are presented in the input:
  - a. The material model has a hydrodynamic volume limit (HVL).
  - b. A failure model is defined.
  - c. DYPARAM, HVLFAIL, YES
2. The hydrodynamic volume limit by default allows for 10% expansion.

**DYPARAM,IMM**

Initial Metric Method Formulation - SOL700

The option allows to specify the IMM method to be used. Used in SOL 700 only.

**Format:**

DYPARAM, IMM, OPTION1, OPTION2, VALUE1, VALUE2

**Example:**

DYPARAM, IMM, ZERO, YES, 1.0E-3, 1.0E-3

| Descriptor | Meaning  |
|------------|--|
| OPTION1    | Flag for element condition under compression. (Character; default=FULL)  |
| FULL       | While elements are under IMM condition, they will carry stresses when under compression.   |
| REDUCED    | While elements are under IMM condition they will carry a reduced stress when under compression. The relative area factor SMDFER is used to reduce the Young's modulus. |
| ZERO       | While elements are under IMM condition they do not carry any compressive stresses. Use material damping to avoid excessive nodal velocities.                           |
| OPTION2    | Flag for recalculation of IMM strain. (Character; default=See remark 2.)   |
| NO or OFF  | Do not recalculate IMM strains during the calculation.   |
| YES or ON  | Recalculate IMM strains during the calculation.  |
| VALUE1     | Start time of recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)   |
| VALUE2     | Times between recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)   |

**Remarks:**

- Method ZERO is best suitable when initially more than a couple of elements with zero or near zero area are present in the model.
- The default for Option2 depends on Option1.

| Option1 | Default for Option2 |
|---------|---------------------|
| FULL    | OFF                 |
| REDUCED | OFF                 |
| ZERO    | ON                  |

- When Option2 is OFF or NO, STREC and DTREC are neglected.

## DYPARAM,INFOBJ

List the Generated BJOINs and Spotwelds - SOL700

Additional information about the BJOIN and spotweld connectivity will be listed in the output file. Used in SOL 700 only.

Format:

DYPARAM, INFOBJ, value

Example:

DYPARAM, INFOBJ, 0.14

| Descriptor | Meaning  |
|------------|--|
| value      | Flag of activation of writing BJOIN information. (Character; default=NO) |
| YES        | Information is issued.   |
| NO         | Information is not issued.   |

Remarks:

1. The information listed is:
  - Grid point pairs forming a BJOIN or a spotweld.
  - BJOINs and spotwelds initially connected.

## DYPARAM,INISTEP

Initial Time Step - SOL700

Defines the time step used at the start of the analysis. Used in SOL 700 only.

**Format:**

DYPARAM, INISTEP, value

**Example:**

DYPARAM, INISTEP, 1.0E-6

| Descriptor | Meaning   |
|------------|---|
| value      | Time step (in analysis time units) used for the first iteration. (Real > 0.0; required) |

**Remarks:**

1. This parameter is required to start an analysis.

**DYPARAM,LIMCUB**

## Contact Cube Sort Algorithm - SOL700

Defines the maximum number of cubes used to sort the grid points in a contact definition. Used in SOL 700 only.

**Format:**

DYPARAM, LIMCUB, value

**Example:**

DYPARAM, LIMCUB, 2300

| Descriptor | Meaning   |
|------------|---|
| value      | Maximum number of cubes. (Integer > 0; default=200) |

**Remarks:**

1. Each slave node has to search for master nodes that are close enough to have potential contact. It is too expensive to have each slave node check each master node. To limit the number of checks, the space in which the nodes reside is subdivided into cubes. This subdivision is done so that the slave nodes have to check only the master nodes in their own cube and those in the neighboring cubes. The maximum number of cubes used to subdivide the space is equal to the value of LIMCUB.

**DYPARAM,MATRMERG**

Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly. Used in SOL 700 only.

**Format:**

```
DYPARAM, MATRMERG, FR<id1>, MR<id2>, MR<id3>, FR<id4>,...  
DYPARAM, MATRMERG, AUTO
```

**Example:**

```
DYPARAM, MATRMERG, FR1, MR2, MR6, MR7, FR4, MR8
```

| Descriptor         | Meaning  |
|--------------------|--|
| FR<id1>            | Name of the new FULLRIG assembly. (Character; required.)   |
| MR<idi> or FR<idi> | Names of MATRIG and/or RBE2-FULLRIG rigid bodies merged into a new FULLRIG assembly with name FR<id1>. No names can be supplied for the AUTO option. |
| AUTO               | Automatic merge. See remark 2. (Character; required.)  |

**Remarks:**

1. FR<id1> must be a nonexisting RBE2-FULLRIG. The properties of FR<id1> (as mass, center of gravity, and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. Rigid body output can be asked for FR<id1>, and loads or rigid body constraints can be applied to FR<id1>. The other MATRIGs and RBE2-FULLRIGs mentioned on the MATRMERG entry disappear after they have been merged.
2. Instead of supplying rigid body names, the AUTO option can be used. After all the normal DYPARAM,MATRMERG and DYPARAM,MATRMRG1 entries have been applied, a DYPARAM,MATRMERG,AUTO merges all the resulting MATRIGs and RBE2-FULLRIGs which have common grid points into a new rigid assembly called FM<id>, where the id is a new FM number starting from 1. As it is not known at the start of an analysis how many FM assemblies will be created, no rigid body output can be asked for FM<id>, and no constraints or loads can be applied to FM<id>. The MATRIGs and RBE2-FULLRIGs, which have been merged by the AUTO option into a new FM<id> assembly, disappear.
3. To supply predefined properties for the merged assembly, DYPARAM,MATRMRG1 can be used, where the first rigid body mentioned on the entry must be an existing RBE2-FULLRIG or MATRIG.

**DYPARAM,MATRMRG1**

Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties. Used in SOL 700 only.

**Format:**

```
DYPARAM,MATRMRG1,MR<id1>,MR<id2>,MR<id3>,FR<id4>,...  
DYPARAM,MATRMRG1,FR<id1>,MR<id2>,MR<id3>,FR<id4>,...
```

**Example:**

```
DYPARAM,MATRMRG1,MR1,MR2,MR6,MR7,FR4,MR8
```

| Descriptor         | Meaning  |
|--------------------|--|
| MR<id1> or FR<id1> | Name of the existing MATRIG or FULLRIG assembly (must be an existing one). (Character; required.)  |
| MR<idi> or FR<idi> | Names of MATRIG and/or RBE2-FULLRIG rigid bodies, which are merged with the existing MR<id1> or FR<id1> into a new MATRIG assembly, with name MR<id1> or FR<id1>. (Character; required.) |

**Remarks:**

1. MR<id1> or FR<id1> must be an existing MATRIG or RBE2-FULLRIG, respectively. For a FULLRIG, the properties of FR<id1> (as mass, center of gravity and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. For a MATRIG, the mass of MR<id1> is either the predefined mass on the MATRIG (id1) entry or the predefined density on the MATRIG (id1) entry times the total volume of all MATRIG members in the MATRMRG1 entry. The center of gravity and moments of inertia of MR<id1> are either predefined on the MATRIG (id1) entry, or are otherwise computed from the properties of each rigid body on the entry. The other MATRIGs and RBE2 FULLRIGs mentioned on the MATRMRG1 entry disappear after they have been merged.

## DYPARAM,MAXSTEP

Maximum Time Step - SOL700

Defines the maximum allowable time step. Used in SOL 700 only.

**Format:**

DYPARAM, MAXSTEP, value

**Example:**

DYPARAM, MAXSTEP, 1.E-3

| Descriptor | Meaning   |
|------------|---|
| value      | The maximum time step. (Real > 0.0; default=1.0E20) |

**Remarks:**

1. If the time step calculated by solver is greater than MAXSTEP, the time step is set to MAXSTEP.

## DYPARAM,MESHPLN

Mesh Density for Covering Rigid Planes - SOL700

Defines Mesh density for covering rigid planes. Used in SOL 700 only.

**Format:**

DYPARAM, MESHPLN, value

**Example:**

DYPARAM, MESHPLN, 4

| Descriptor | Meaning   |
|------------|---|
| value      | Rigid planes will be meshed with MESHPLN times MESHPLN dummy quad elements.<br>(Integer > 0; default=3) |

**Remarks:**

1. The default is sufficient in most cases.

**DYPARAM,MINSTEP**

Minimum Time Step - SOL700

Defines the minimum time step that causes the analysis to terminate. Used in SOL 700 only.

**Format:**

DYPARAM,MINSTEP,value

**Example:**

DYPARAM,MINSTEP,1.E-6

| Descriptor | Meaning   |
|------------|---|
| value      | When the time step is less than the MINSTEP value, the analysis terminates. (Real > 0.0;<br>default=10% of DYPARAM,INISTEP) |

**Remarks:**

1. When the elements become very distorted, in a poorly designed mesh for example, or when they have endured a very large distortion, the time step may drop dramatically. The analysis continues, however, and a lot of computer resources may be wasted. This option allows you to specify a minimum time step that causes the analysis to terminate.

**DYPARAM,MIXGAS**

## Controls Updating of Gas Fractions - SOL700

Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature. Used in SOL 700 only.

**Format:**

DYPARAM,MIXGAS,value

**Example:**

DYPARAM,MIXGAS, YES

| Descriptor | Meaning  |
|------------|--|
| value      | Flag of MIXGAS. (Character; default=NO)  |
| YES        | The gas constants for the Euler material and any gas bags are recalculated based on temperature and gas composition. |
| NO         | Euler and gas bag gas constants are not recalculated.  |

**Remarks:**

1. This parameter is only defined for use with GBAG gas bag definitions and/or the single-material Euler solver.
2. This parameter can be used in conjunction with INFLTR and INFLHB inflator definitions and with PORHOLE, PERMEAB, PORFGBG, and PERMGBG porosity definitions.
3. By default, DYPARAM,MIXGAS is set to YES if any INFLHB or INFLGAS entries are present.

## DYPARAM,NZEROVEL

Auto Constrain Failed Nodes - SOL700

Set the velocity of a node to zero in case all attached elements have failed. Used in SOL 700 only.

**Format:**

DYPARAM, NZEROVEL, value

**Example:**

DYPARAM, NZEROVEL, YES

| Descriptor | Meaning  |
|------------|--|
| value      | Flag of NZEROVEL. (Character; default=NO)  |
| YES        | Perform check and set the velocity to zero if all attached elements have failed. |
| NO         | Do not perform check.  |

**Remarks:**

1. This parameter applies only to nodes of Lagrangian elements.
2. Specifying NO reduces the CPU overhead time.
3. When the velocity of a node is set to zero, effectively the node is constraint, like an SPC or SPC1.
4. Special attention is necessary for the contact definition. If the failed node is not taken out of the contact, it behaves as a rigid boundary constraint.

**DYPARAM,OLDLAGT**

Use Collapsed Hexahedron Scheme for CTETRA - SOL700

Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements. Used in SOL 700 only.

**Format:**

DYPARAM,OLDLAGT,value

**Example:**

DYPARAM,OLDLAGT,1

| Descriptor | Meaning   |
|------------|---|
| value      | Flag of OLDSLGT. (Integer $\geq 0$ ; default=See remark 1.) |
| 0          | Deactivate collapsed hexahedron scheme                      |
| 1          | Activate  |

**Remarks:**

1. The current default integration scheme for Lagrangian CTETRA elements use linear tetrahedron FE one. It is more consistent (in terms of accuracy) and efficient (both in memory and CPU time) compared with the collapsed hexahedron scheme. The old scheme based on collapsed hexahedron with reduced integration is deactivated. If the old scheme is activated, it is used as default. But, it is still possible to use the new scheme for CTETRA by using separate PSOLID with IN = 1 and ISOP = 1 combination.

**DYPARAM,JWLDET**

Hydro Body Force - SOL700

Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries.

Format:

PARAMJWLDET,OPTION

Example:

PARAM,JWLDET,NOLINK

| Descriptor | Meaning  |
|------------|--|
| LINK       | Multiple denotations with EOSJWL are LINKED.<br>The detonation wave of one explosive can ignite another explosive.       |
| NOLINK     | Multiple denotations with EOSJWL are NOT LINKED.<br>The detonation wave of one explosive cannot ignite another explosive |
| Default    | NOLINK   |

Remark:

1. Option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model.
2. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

**DYPARAM,LIMITER**

Original Roe-Type Euler Solver Scheme - SOL700

Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe. Used in SOL 700 only.

**Format:**

DYPARAM,LIMITER,TYPE,OPTION

**Example:**

DYPARAM,LIMITER,ROE

| Descriptor | Meaning   |
|------------|---|
| TYPE       | Type of scheme. (Character; Default = ROE)                    |
|            | ROE      Roe solver for single hydrodynamic materials.        |
| OPTION     | blank     Second order in space. (Character; Default = blank) |
|            | NONE    First order in space.                                 |

**Remarks:**

1. By default, when the parameter is not set, the solver is used that is defined on the PEULER or the PEULER1 entry. In case “2ndOrder” or “1stOrder” was defined on the PEULER or PEULER1 entry, the parameter setting has no effect.
2. By default, second order spatial accuracy is used. The temporal accuracy is automatically defined according to the spatial accuracy that you select.
3. Note that 2<sup>nd</sup> order spatial accuracy in the Roe solver involves the internal flow field only. We recommend that you use the full 2<sup>nd</sup> order improved fluid-and gas Euler solver. You can activate the improved solver by putting the “2ndOrder” field on the PEULER or PEULER1 entry.
4. When type ROE is defined, no void elements are allowed and it cannot be used in combination with EOSJWL. Also options concerning air-bag analyses are not supported.
5. For more details on fluid- and gas Euler solvers, refer to the Getting Started and the Theory Manuals.

## DYPARAM,PARALLEL

Parallel Execution Information - SOL700

The option allows you to gather information on the parallel section. Used in SOL 700 only.

**Format:**

DYPARAM, PARALLEL, INFPAR, value

**Example:**

DYPARAM, PARALLEL, INFPAR, ON

| Descriptor | Meaning   |
|------------|---|
| INFPAR     | A report is written on the actual amount of work done at the reported parallel levels.<br>(Character; required) |
| Value      | Flag for INFPAR. (Character; default=OFF)   |
|            | ON              Activate writing information.   |
|            | OFF             Deactivate writing information.   |

**Remarks:**

1. A summary on the parallel operation when using the shared-memory mode can be requested by including a DYPARAM,PARALLEL,INFPAR,ON entry in the input file.
2. Currently, the information on the parallel sections is available for the shell solver only.

**DYPARAM,PLCOVCUT****Pressure Cut Off Time - SOL700**

Defines time when PLCOVER is cut off. Used in SOL 700 only.

**Format:**

DYPARAM, PLCOVCUT, value

**Example:**

DYPARAM, PLCOVCUT, 3.E-3

| Descriptor | Meaning   |
|------------|---|
| value      | If there are one or more COUPLE definitions with a PLCOVER specified on the COUOPT entry, a cut off is applied to the PLCOVER until time = PLCOVCUT. From time = 0 to time = PLCOVCUT, the PLCOVER is cut off to the pressure in the intersected Eulerian element.<br>For times greater than PLCOVCUT, the full PLCOVER is applied to the coupling surface.<br>This parameter is useful in air-bag analyses, where PLCOVER is used to model the environment pressure. During the early stages of the deployment of the air bag, the pressure inside the bag may drop. Applying the full PLCOVER may lead to an unstable deployment of the air bag. (Real $\geq 0.0$ ; default=0). |

**Remarks:**

1. See also the COUPLE and COUOPT Bulk Data entries.

## DYPARAM,PMINFAIL

Switches Failure at Spall Limit - SOL700

Defines Lagrangian solid element failure on reaching the spall limit. Used in SOL 700 only.

**Format:**

DYPARAM, PMINFAIL, value

**Example:**

DYPARAM, PMINFAIL, YES

| Descriptor | Meaning                                   |
|------------|---|
| VALUE      | Flag of PMINFAIL. (Character; default=NO) |
| YES        | Element failure on spall limit            |
| NO         | No element failure on spall limit         |

**Remarks:**

1. Lagrangian elements (CHEXA) that have a material definition with a failure model will fail when the parameter is set to YES and the spall limit (minimum pressure) is reached, even when the other failure criterion is not yet reached.
2. The spall limit is set on the PMINC entry.

## DYPARAM,RBE2INFO

Lists MATRIG and RBE2 Grid Points - SOL700

The grid points attached to MATRIG and RBE2 assemblies are listed to the output file. Used in SOL 700 only.

**Format:**

DYPARAM, RBE2INFO, value

**Example:**

DYPARAM, RBE2INFO, GRIDON

| Descriptor | Meaning  |
|------------|--|
| VALUE      | Flag of PBE2INFO. (Character; default=GRIDOFF) |
| GRIDON     | Information is issued C                        |
| GRIDOFF    | No information is issued                       |

## DYPARAM,RHOCUT

Global Density Cutoff Value - SOL700

Defines the minimum density for all Eulerian elements. Used in SOL 700 only.

**Format:**

DYPARAM, RHOCUT, value

**Example:**

DYPARAM, RHOCUT, 1.E-10

| Descriptor | Meaning   |
|------------|---|
| value      | Density cutoff. (Real > 0.0; default=See remark 4.) |

**Remarks:**

1. Any Eulerian element with a density less than RHOCUT is considered to be empty. All of its variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than RHOCUT, then no transport is done.
  - b. If the density of element A after transport is less than RHOCUT, then all of the mass is transported to element B.
3. A reasonable value of RHOCUT is 1.E-5 times the initial density.
4. If only RHOCUT is defined, all Eulerian elements use the RHOCUT value as cutoff density. If RHOCUT is omitted, all Eulerian elements use a cutoff density automatically set to 1.E-5 times a characteristic density. For single-material Eulerian elements, this characteristic density is the reference density.

**DYPARAM,RJSTIFF****Rigid-joint Stiffness - SOL700**

Defines the stiffness of a rigid joint. Used in SOL 700 only.

**Format:**

DYPARAM, RJSTIFF, value

**Example:**

DYPARAM, RJSTIFF, 100.

| Descriptor | Meaning  |
|------------|--|
| value      | Multiplication factor for the stiffness of all rigid joints. (Real > 0.0; default=1.0) |

**Remarks:**

1. The absolute stiffness of rigid joints is calculated automatically by solver. The stiffness of joints is taken so that a stable solution is guaranteed. The stiffness calculation takes into account the fact that a rigid body can be constrained by more than one joint.
2. This parameter can be used to increase or decrease the stiffness of the joints. Care must be taken because too high a value may lead to an unstable calculation.

**DYPARAM,SCALEMAS**

Mass Scaling Definition - SOL 700

Defines the activation of mass scaling.

**Format:**

DYPARAM, SCALEMAS, DTMIN, MXPERC, STEPS.

**Example:**

DYPARAM, SCALEMAS, 1.0E-6, 100.0, 1

| Descriptor | Meaning  |
|------------|--|
| DTMIN      | Minimum allowable time step. (Real > 0.0; required)  |
| MXPERC     | Maximum percentage of added numerical mass with respect to original mass. (Real > 0.0; required) |
| STEPS      | Number of steps. (Integer > 0; required)   |

**Remarks:**

1. Numerical mass is added to all Lagrangian solid, triangular, quadrilateral, rod, bar and beam elements such that its time step never becomes less than:

$$dt = STEPFACT * DTMIN.$$

Where,

dt = time step of calculation

STEPFACT = time step safety factor (see PARAM,STEPFACT)

DTMIN = value specified on the PARAM,SCALEMAS entry

If the added mass of a certain element exceeds the maximum percentage (MXPERC) of its original mass, no more mass will be added, and subsequently, the time step may decrease again.

2. The value of STEPS determines the checking frequency against the mass scaling criterion; the check is done for every defined number of STEPS. STEPS = 1 is recommended.
3. By requesting MSMASS in an ELOUT output request, the ratio of scaled mass to original mass of the elements can be retrieved. By making fringe plots of this parameter, a check can be made if mass has not been added in a critical area.
4. See *MSC Nastran Explicit Nonlinear (SOL 700) User's Guide*, Chapter 4: Special Modeling Techniques, [Mass Scaling](#) Definition for instructions on how to use this entry

## DYPARAM,SHELLFRM

Sets the Default of the Shell Formulation - SOL700

Sets the default for the shell formulation for quadrilateral elements. Used in SOL 700 only.

Format:

DYPARAM, SHELLFORM, value

Example:

DYPARAM, SHELLFORM, BLT

| Descriptor | Meaning   |
|------------|---|
| value      | Flag for default of shell formulation. (Character; default=KEYHOFF) |
| BLT        | The shell-formulation default is BLT.                               |
| KEYHOFF    | The shell-formulation default if KEYHOFF.                           |

Remarks:

1. The DYPARAM,SHELLFORM changes the default formulation for quadrilateral shell elements. All shell properties entries that do not explicitly define the formulation, use the default as specified on the DYPARAM entry.
2. Triangular shell elements have only one formulation (C0-TRIA). Therefore, the DYPARAM is ignored for triangular elements.

## DYPARAM,SHELMSYS

Shell Element System Definition - SOL700

Defines the shell element system for the BLT shells. Used in SOL 700 only.

**Format:**

DYPARAM, SHELMSYS, value

**Example:**

DYPARAM, SHELMSYS, SIDE21

| Descriptor | Meaning   |
|------------|---|
| value      | Type of shell element system. (Character; default=MIDSIDES) |
| SIDE21     | x-axis along side21.  |
| MIDSIDES   | x-axis connecting midpoints.                                |

**Remarks:**

1. SIDE21 puts the x-axis along side21 of the element, whereas MIDSIDES puts the x-axis along the vector connecting the midpoints of the side14 and side32.
2. Using the SIDE21 option for the BLT shell will result in the same Belytschko-Lin-Tsay implementation as BELY.

**DYPARAM,SHPLAST**

## Type of Plane-Stress Plasticity for Shells - SOL700

Specifies the type of calculation used to determine the plane-stress plasticity method for shells. Used in SOL 700 only.

**Format:**

DYPARAM, SHPLAST, value

**Example:**

DYPARAM, SHPLAST, VECT

| Descriptor | Meaning   |
|------------|---|
| value      | Type of Plain-stress plasticity for shells. (Character; default=ITER) |
| RADIAL     | Noniterative, approximate radial return.                              |
| VECT       | Iterative, vectorized with three iterations.                          |
| ITER       | Nonvectorized iterations.   |

**Remarks:**

1. The RADIAL approach does not require iterations and, therefore, is the most efficient. It is, however, an approximation.
2. The other two approaches iterate to find the solution. ITER is the best since it takes as many iterations as are necessary. On vector machines, such as CRAY, this is inefficient since it cannot be vectorized. VECT always performs three vectorized iterations, which is more efficient. However, three iterations may not be enough, and inaccuracies could occur.

**DYPARAM,SHSTRDEF**

## Composite Shell Stress and Strain Output Definition - SOL700

Specifies the default coordinate system for the stress and strain output of composite shells. Used in SOL 700 only.

**Format:**

DYPARAM, SHSTRDEF, value

**Example:**

DYPARAM, SHSTRDEF, ELEM

| Descriptor | Meaning   |
|------------|---|
| value      | Type of output. (Character; default=FIBER)                          |
| FIBER      | Stresses and strains are output in the fiber and matrix directions. |
| ELEM       | Stresses and strains are output in the element coordinate system.   |

**Remarks:**

1. The default setting can be overruled per property on a PCOMPA entry on the STRDEF field.

## DYPARAM,SHTHICK

## Shell-Thickness Modification Option - SOL700

Specifies whether or not the thickness of the shell changes with membrane straining. Used in SOL 700 only.

**Format:**

DYPARAM, SHTHICK, value

**Example:**

DYPARAM, SHTHICK, YES

| Descriptor | Meaning  |
|------------|--|
| value      | Flag of shell thickness modification. (Character; default=YES) |
| YES        | Shell thickness is modified according to the membrane strain.  |
| NO         | Shell thickness is constant.                                   |

**Remarks:**

1. The YES option gives a true large-strain shell but requires some extra computation.
2. The NO option should give adequate results as long as the membrane strains are not very large (i.e., not more than 5–10%).
3. This option applies to all the formulations of the shell elements, except for the PCOMP. The thickness of PCOMP shell elements will always remain constant.

## DYPARAM,SLELM

Store Shell Sublayer Variables - SOL700

Defines whether shell sublayer variables are to be stored in the element arrays. Used in SOL 700 only.

**Format:**

DYPARAM, SLELM, value

**Example:**

DYPARAM, SLELM, NO

| Descriptor | Meaning  |
|------------|--|
| value      | Flag of storing shell sublayer variables. (Character; default=YES) |
| YES        | Store as an element variable.                                      |
| NO         | Do not store as an element variable.                               |

**Remarks:**

1. This parameter applies only to shell elements.
2. The shell sublayer variables are primarily stored in sublayer arrays. They can be copied into the element arrays only for specific output purposes.
3. Specifying NO reduces the CPU overhead time.
4. Irrespective of the entry on this parameter, sublayer variables are accessible in the sublayer arrays. For example, requesting TXX1 retrieves the stress from the element array, whereas TXX01 retrieves it from the sublayer arrays.

**DYPARAM,SMP,BATCHSIZ****SMP Batchsize and CPULOOP Definition - SOL700**

Define batch size and number of CPU loops, per entity type. Used in SOL 700 only.

**Format:**

DYPARAM, SMP, <entity>, BATCHSIZ, <value1>, CPULOOPS, <value2>

**Example:**

DYPARAM, SMP, BEAMS, BATCHSIZ, AUTOMAT  
DYPARAM, SMP, BEAMS, BATCHSIZ, 256

| Descriptor | Meaning   |
|------------|---|
| ENTITY     | Define the batchsize for the following entities. (Character; required)  |
| BEAMS      | Beam elements   |
| SHELLS     | Shell elements  |
| SOLIDs     | Solid elements  |
| CONTACT    | Contacts  |
| VALUE1     | The batchsize for this entity (Integer number) or AUTOMAT (character). See Remarks 1.and 3. (Character or Integer >0; default = 63) |
| VALUE2     | If value1 is defined as AUTOMATIC, the user may define the number of loops per CPU<br>See Remark 2.(Integer > 0; default=2)         |

**Remarks:**

1. The batchsize may be defined per entity type. The SMP parts uses the batchsize to assign a certain number of batches to each CPU. Example:
  - Batchsize = 63 (default)
  - Number of Beam elements = 580
  - Number of CPUs = 4

As a results,

- Needs to process 10 batches ( $9 \times 63 + 1 \times 13$ )
- To spread this evenly over four CPUs, a solver increases this to 12 batches because then each CPU can process 3 batches.
- As a result, the batch size is recalculated for 12 batches to: batchsize = 49
- This means that:

CPU# 1,2,3 process three batches of 49 elements each ( $3 \times 3 \times 49 = 441$  elements)

CPU# 4 processes two batches of 49 elements and one batch of 41

It is clear that the user-defined batchsize may not be used ultimately. The reason for this is that each CPU is allocated the same number of batches in order to optimize the work done by each processor.

2. When the batchsize is set to AUTOMAT, a solver computes an initial batchsize based on the number of batches processed by each CPU. By default, processes two batches per CPU.

In the example above, this means that based on:

- Batchsize = AUTOMAT
- Number of batches per CPU = 2 (default)
- Number of Beam elements = 580
- Number of CPUs = 4

As a result,

- Computes an initial batchsize of 72 ( $=580/(4*2)$ )
- Increases this batchsize to a higher number to make sure the number of batches fits in the fixed number of CPU's: batchsize =  $72 + (580 - 4*2*72) = 72 + 4 = 76$  elements
- Based on this initial batchsize, the number of batches would be:  $1 + (580-1)/76 = 8$  batches
- This fits nicely on four CPU's as expected
- The CPUs process:

CPU# 1,2,3 processes two batches of 76 elements each ( $3 \times 2 \times 76 = 456$  elements)

CPU# 4 processes one batch of 76 elements and 1 batch of 48 elements.

3. By default, a solver does not use the AUTOMAT batchsize algorithm. Testing has shown that an initial batchsize of 63 yields the fastest results. This may vary depending on the problem simulated and the number of elements and may be worth adjusting.

The reason for this may be found in the fact, that with many elements in the model, the number of elements per batch increases significantly when the AUTOMAT batchsize algorithm is used. Each CPU will have to process two batches (by default) with many elements. Based on the cache memory, this is usually not optimal for a CPU processor. Shorter batchsizes (< 128) will do a better job.

**DYPARAM,SMP,CPUINFO****SMP CPU Information - SOL700**

Define the CPU information per entity type. Used in SOL 700 only.

**Format:**

DYPARAM, SMP, <entity>, CPUINFO, <value1>

**Example:**

DYPARAM, SMP, CONTACT, CPUINFO, ON

| Descriptor | Meaning  |
|------------|--|
| ENTITY     | Define the CPUINFO for the following entities. (Character; required) |
| BEAMS      | Beam elements  |
| SHELLS     | Shell elements   |
| SOLIDs     | Solid elements   |
| CONTACT    | Contacts   |
| MEMBR      | Membrane elements  |
| EULT9      | Eulerian Roe Solver  |
| FSCOUP     | Fast coupling algorithm  |
| ALL        | Output for all of the above  |
| VALUE1     | Flag for writing CPU information. (Character; required)              |
| ON         | Ask for parallel CPU time information C                              |
| OFF        | No CPU time information (default)                                    |

**Remarks:**

1. By default, a solver does not give any SMP CPU timings per entity. In order to judge the efficiency of the different algorithms, this parameter may be used to judge the speedup acquired by processing on multiple CPUs.

## DYPARAM,SNDLIM

Sound Speed Minimum Value - SOL700

Defines the minimum value for the speed of sound. Used in SOL 700 only.

**Format:**

DYPARAM, SNDLIM, value

**Example:**

DYPARAM, SNDLIM, 1.E-6

| Descriptor | Meaning   |
|------------|---|
| value      | Minimum value of speed of sound. (Real > 0.0; default=1.0E-3) |

**Remarks:**

1. This parameter is used to avoid the possibility of division by zero in the time step calculation.
2. SNDLIM has the units of velocity.

**DYPARAM,SPREMAP****1-D Spherical Symmetric Euler Archive Remap - SOL700**

Allows a 1-D spherical symmetric Euler archive importation to a 3-D simulation. Used in SOL 700 only.

**Format:**

DYPARAM, SPREMAP, X0, Y0, Z0, range

**Example:**

DYPARAM, SPREMAP, 0.0, 0.5, 0.5

| Descriptor | Meaning   |
|------------|---|
| X0, Y0, Z0 | X, Y, Z coordinate of the point at which the 1-D spherical symmetric mesh is remapped. (Real; default=0.0)  |
| RANGE      | Only material whose distance to (X0,Y0,Z0) is smaller than “range” will be initialized with the 1-D spherical symmetric Euler archive. (Real > 0.0; default = 1.0E20) |

**Remarks:**

1. Since 1-D spherical symmetric simulations run much faster than 3-D simulation, it can save much CPU time to do the first part of the simulation with a 1-D axial symmetric mesh. Afterwards, the 1-D spherical symmetric Euler archive is imported into the 3-D simulation. By default, the 1-D spherical symmetric archive will not be expanded in 3-D. To enable this expansion, DYPARAM,SPREMAP has to be used. It is useful for blast wave simulations. The 1-D spherical symmetric simulation has to be terminated before the blast wave approaches any 3-D structure.
2. This import of Euler archives is done by means of the EID option in the pth file.
3. To generate axial symmetric meshes DYPARAM,SPHERSYM can be used.
4. The remap of 1-D spherical symmetric to axial symmetric Euler meshes is supported.

**DYPARAM,SPHERSYM**

## Spherical Symmetric Analyses - SOL700

Enables an efficient and accurate 1D spherical symmetric solution for Eulerian materials. A much larger time step becomes possible by basing the time step only on the mesh-size in radial direction. Used in SOL 700 only.

**Format:**

```
DYPARAM, SPHERSYM, MESHTYPE, RADAXIS, PHI
```

**Example:**

```
PARAM, SPHERSYM, RECT, X, 2.0
```

| Descriptor | Meaning   |
|------------|---|
| MESHTYPE   | Two types of Euler meshes are supported. (Character; required)  |
| SPHERIC    | Axial symmetric meshes  |
| RECT       | Rectangular meshes  |
| RADAXIS    | Radial Axis. (Character; required.)   |
| X          | Global x direction  |
| Y          | Global y direction  |
| Z          | Global z direction  |
| PHI        | Only used for MESHTYPE = RECT. Used to creates a 1d Spherical mesh with angles +PHI/2 and -PHI/2. (Real $\geq 0$ ; default=0.0) |

**Remarks:**

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.
2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one row of elements can be used. Using the angle specified by PHI this Euler mesh is mapped into a 1D spherical symmetric mesh.
3. The Euler mesh has to consist of one row of elements.
4. In the time step computation only the mesh-size in radial direction will be taken into account.

**DYPARAM,STRNOUT**

Shell Sublayer Strain Output - SOL700

Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output. Used in SOL 700 only.

**Format:**

DYPARAM, STRNOUT, value

**Example:**

DYPARAM, STRNOUT, YES

| Descriptor | Meaning   |
|------------|---|
| value      | Flag for writing strain results. (Character; default=YES) |
| YES        | Save  |
| NO         | Do not save.  |

**Remarks:**

1. A limited output set saves memory.
2. Perfectly elastic materials only have the limited output set.
3. Total strain output for shell composite materials can be requested from the PCOMPA Bulk Data entry.

## DYPARAM,TOLCHK

## Tolerance for Fast Coupling - SOL700

Fast coupling requires that Euler elements are orthogonal. This means that normals of Euler element faces have to be in a coordinate direction. In practice there can be small errors in the geometry of the Euler element and Euler face normals do not always exactly point into a coordinate direction. Small deviations from the coordinate direction do not give problems and are allowed by fast coupling.

To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK , write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14. Used in SOL 700 only.

**Format:**

DYPARAM, TOLCHK, value

**Example:**

DYPARAM, TOLCHK, 1e-10

| Descriptor | Meaning  |
|------------|--|
| value      | Tolerance in accepting faces by fast coupling. (Real > 0.0; required.) |

**DYPARAM,VDAMP**

## Dynamic Relaxation Parameter - SOL700

Controls the global damping in the dynamic relaxation. Used in SOL 700 only.

**Format:**

DYPARAM, VDAMP , value

**Example:**

DYPARAM, VDAMP , 0.001

| Descriptor | Meaning   |
|------------|---|
| value      | Dynamic relaxation parameter. (Real $\geq 0.0$ ; default=0.0) |
| 0.0        | No dynamic relaxation.  |

**Remarks:**

1. The dynamic relaxation parameter is connected to the system natural frequency, ?, as  $?=s??t$ , where s denotes a percentage of critical damping. The damping occurs by factoring the velocities every time step as follows:

$$F_1 = (1-\beta)/(1+\beta)$$

$$F_2 = 1/(1+\beta)$$

$$v^{n+1/2} = F_1 v^{n-1/2} + F_2 a^n \beta t^n$$

where v is the velocity, a is the acceleration, and ? is the dynamic relaxation parameter.

## DYPARAM,VELMAX

Maximum Velocity - SOL700

Defines the maximum velocity in Eulerian and Lagrangian meshes. Used in SOL 700 only.

**Format:**

DYPARAM, VELMAX, value, option

**Example:**

DYPARAM, VELMAX, 1.E6

| Descriptor | Meaning   |
|------------|---|
| VALUE      | Maximum velocity. (Real > 0.0; default=1.0E10)  |
| OPTION     | Flag for removing Euler elements. (Character; default=YES)  |
| YES        | Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.        |
| NO         | Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity. |

**Remarks:**

1. For Eulerian elements, VELMAX is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAX. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAX.
2. Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
3. Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
4. VELMAX must be greater than the minimum velocity specified by PARAM,VELCUT.

**DYPARAM,VELMAX1****Maximum Translational and Angular Velocity - SOL700**

Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes. Used in SOL700 only.

**Format:**

DYPARAM, VELMAX1, VELMAXT, option, VELMAXA

**Example:**

DYPARAM, VELMAX1, 1.E6

| Descriptor | Meaning  |   |
|------------|--|---|
| VELMAXT    | Maximum translational velocity. (Real > 0.0; default=1.0E20) |   |
| OPTION     | Flag for removing Euler elements. (Character; default=YES)   |   |
|            | YES  | Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.        |
|            | NO   | Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity. |
| VELMAXA    | Maximum angular velocity. (Real > 0.0; default=1.0E20)       |   |

**Remarks:**

- For Eulerian elements, VELMAXT is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAXT. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAXT.
- VELMAXA is used to limit the angular velocity of grid points. By default, the angular velocities are not limited. In general, there is no need to limit angular grid point velocities.
- Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
- Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
- VELMAXT must be greater than the minimum velocity specified by PARAM, VELCUT.
- DYPARAM, VELMAX, and DYPARAM, VELMAX1 can be used in the same input deck. The values set by DYPARAM, VELMAX1 will overrule the values set by DYPARAM, VELMAX.

For example, when using

DYPARAM, VELMAX, 1000, YES

```
DYPARAM,VELMAX1,,YES,1e+10
```

In the OUT file shows:

```
%I-INIT 02-p3_set_velmaxat, , ,(13),  
SUMMARY VELMAX SETTINGS:  
MAXIMAL VALUE TRANSLATIONAL VELOCITY = 0.100000E+04  
MAXIMAL VALUE ANGULAR VELOCITY = 0.100000E+11  
REMOVE EULERIAN MASS WHEN VELOCUITY EXCEEDS LIMIT: YES
```

**DYPARAM,VISCPLAS**

Use Overstress Formula to Update Strain-rate Dependent Plasticity - SOL700

Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material. Used in SOL 700 only.

**Format:**

DYPARAM, VISCOPLAS, value

**Example:**

DYPARAM, VISCOPLAS, 1

| Descriptor | Meaning   |
|------------|---|
| VALUE      | Flag of updating overstress formula. (Integer $\geq 0$ ; default=0) |
| 0          | Use scaling-up scheme   |
| 1          | Use overstress formula  |

**Remarks:**

1. The strain rate dependent plasticity is normally calculated by scaling up the basic yield stress without strain rate effect. Then the trial stresses are mapped back to the scaled-up yield surface. This algorithm may lead to premature instability. Another technique is to calculate the so-called viscous-plastic strain rate using “overstress” formula. And then the stresses are updated based on this viscous-plastic strain. This technique seems to be more stable than the previous one. For shell elements, this option works when combined with DYPARAM,SHPLAST,RADIAL. Only MATEP and Johnson-Cook models are supported. For solid elements, this option works only for MATEP.
2. For shell elements, DYPARAM,VISCOPLAS,1, in combination with DYPARAM,SHPLAST,RADIAL, will use consistent plane stress plasticity algorithm both for strain rate dependent and independent plasticity. This new algorithm is more accurate than 3-D approach.

**ECHOOFF**

## Deactivate Printed Echo

Marks the point or points in the input file to deactivate printed echo of the input file.

Format:

| 1       | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|---|---|---|---|---|---|---|---|----|
| ECHOOFF |   |   |   |   |   |   |   |   |    |

Example:

|         |  |  |  |  |  |  |  |  |  |
|---------|--|--|--|--|--|--|--|--|--|
| ECHOOFF |  |  |  |  |  |  |  |  |  |
|---------|--|--|--|--|--|--|--|--|--|

Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\), 310](#).
2. The companion to this entry is the ECHOON entry.
3. The ECHOOFF command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended: ECHO=UNSORT.

**ECHOON****Activate Printed Echo**

Marks the point or points in the input file to activate printed echo of the input file.

**Format:**

| 1      | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---|---|---|---|---|---|---|---|----|
| ECHOON |   |   |   |   |   |   |   |   |    |

**Example:**

|        |  |  |  |  |  |  |  |  |  |
|--------|--|--|--|--|--|--|--|--|--|
| ECHOON |  |  |  |  |  |  |  |  |  |
|--------|--|--|--|--|--|--|--|--|--|

**Remarks:**

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\), 310](#).
2. The companion to this entry is the ECHOOFF entry.
3. The ECHOON command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended:  
ECHO=UNSORT.

**EIGB****Buckling Analysis Set**

Defines data needed to perform buckling analysis.

**Format:**

| 1    | 2    | 3      | 4  | 5  | 6   | 7   | 8   | 9 | 10 |
|------|------|--------|----|----|-----|-----|-----|---|----|
| EIGB | SID  | METHOD | L1 | L2 | NEP | NDP | NDN |   |    |
|      | NORM | G      | C  |    |     |     |     |   |    |

**Example:**

|      |    |     |     |     |   |   |   |  |  |
|------|----|-----|-----|-----|---|---|---|--|--|
| EIGB | 13 | INV | 0.1 | 2.5 | 2 | 1 | 1 |  |  |
|      |    | MAX |     |     |   |   |   |  |  |

| Descriptor | Meaning   |   |
|------------|---|---|
| SID        | Set identification number. (Unique Integer > 0)   |   |
| METHOD     | Method of eigenvalue extraction. (Character: “INV” for inverse power method or “SINV” for enhanced inverse power method.) |   |
| L1, L2     | Eigenvalue range of interest. (Real, L1 < L2)   |   |
| NEP        | Estimate of number of roots in positive range not used for METHOD = “SINV”. (Integer > 0)                                 |   |
| NDP, NDN   | Desired number of positive and negative roots. (Integer>0; Default = 3*NEP)   |   |
| NORM       | Method for normalizing eigenvectors. (Character: “MAX” or “POINT”; Default = “MAX”)                                       |   |
|            | MAX   | Normalize eigenvectors to the unit value of the largest component in the analysis set. (Default).   |
|            | POINT   | Normalize eigenvectors to the unit value of the component defined in G and C fields. The value for NORM defaults to MAX if the defined component is zero. |
| G          | Grid or scalar point identification number. Required only if NORM=“POINT”. (Integer > 0)                                  |   |
| C          | Component number. Required only if NORM=“POINT” and G is a geometric grid point. (1 ≤ Integer ≤ 6)                        |   |

**Remarks:**

1. The EIGB entry must be selected with the Case Control command METHOD = SID.
2. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.

3. The continuation entry is optional. If the continuation is not specified, than NORM = “MAX” normalization is performed.
4. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
5. The SINV method is an enhanced version of the INV method. It uses Sturm sequence techniques to ensure that all roots in the range have been found. It is generally more reliable and more efficient than the INV method.
6. Convergence is achieved at  $10^{-6}$ .
7. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the L1 and L2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If NDP and NDN are set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

**EIGC****Complex Eigenvalue Extraction Data**

Defines data needed to perform complex eigenvalue analysis.

**Format:**

| 1    | 2   | 3      | 4    | 5 | 6 | 7 | 8   | 9 | 10 |
|------|-----|--------|------|---|---|---|-----|---|----|
| EIGC | SID | METHOD | NORM | G | C | E | ND0 |   |    |

The following continuation is repeated for each desired search region. (J = 1 to n, where n is the number of search regions.)

|  |         |          |         |         |    |     |     |  |  |
|--|---------|----------|---------|---------|----|-----|-----|--|--|
|  | ALPHAAJ | OMEGA AJ | ALPHABJ | OMEGABJ | LJ | NEJ | NDJ |  |  |
|--|---------|----------|---------|---------|----|-----|-----|--|--|

**Alternate Format for Continuation Entry for Block Complex Lanczos:**

|  |         |          |        |        |        |  |     |  |  |
|--|---------|----------|--------|--------|--------|--|-----|--|--|
|  | ALPHAAJ | OMEGA AJ | MBLKSZ | IBLKSZ | KSTEPS |  | NJi |  |  |
|--|---------|----------|--------|--------|--------|--|-----|--|--|

**Examples:**

|      |    |      |  |  |  |  |   |  |  |
|------|----|------|--|--|--|--|---|--|--|
| EIGC | 14 | CLAN |  |  |  |  |   |  |  |
|      |    | +5.6 |  |  |  |  | 4 |  |  |
| EIGC | 15 | HESS |  |  |  |  | 6 |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Unique Integer > 0)  |
| METHOD     | Method of complex eigenvalue extraction. (Character: "INV", "HESS", or "CLAN")   |
| INV        | Inverse power.   |
| HESS       | Upper Hessenberg. For linear perturbation of ANALYSIS= DCEIG with large displacement, please don't use HESS. See Remarks 2. and 3.   |
| CLAN       | Complex Lanczos. For linear perturbation of ANALYSIS= DCEIG with large displacement, CLAN is recommended. See Remark 8.  |
| IRAM       | Implicitly Restarted Arnoldi Method. See Remark 13.  |
| NORM       | Method for normalizing eigenvectors. See Remark 7. (Character: "MAX" or "POINT"; Default = "MAX")  |
| MAX        | Normalize the component having the largest magnitude to a unit value for the real part and a zero value for the imaginary part.  |
| POINT      | Normalize the component defined in fields 5 and 6 to a unit value for the real part and a zero value for the imaginary part. The value for NORM defaults to "MAX" if the magnitude of the defined component is zero. |
| G          | Grid or scalar point identification number. Required if and only if NORM = "POINT". (Integer > 0)  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component number. Required if and only if NORM = "POINT" and G is a geometric grid point. ( $0 \leq \text{Integer} \leq 6$ )   |
| E          | Convergence criterion. (Real $\geq 0.0$ . Default values are: $10^{-4}$ for METHOD = "INV", $10^{-15}$ for METHOD = "HESS", E is machine dependent for METHOD = "CLAN".) |
| MBLKSZ     | Maximum block size. See Remark 11. (Default = 7; Real $\geq 0.0$ )   |
| IBLKSZ     | Initial block size. See Remark 11. (Default = 2; Real $\geq 0.0$ )   |
| KSTEPS     | Frequency of solve. (Default = 5; Integer $> 0$ )  |

Table 9-11 Relationship Between METHOD Field and Other Fields

| Field  | METHOD Field   |   |  |
|--|--|---|--|
|  | HESS   | INV   | CLAN   |
| NDj (Integer $> 0$ )   | Desired number of eigenvalues and eigenvectors. (No Default) | Desired number of roots and eigenvectors in j-th search region.<br>(Default = $3 * NEj$ ) | Desired number of roots and eigenvectors to be extracted at j-th shift point. (No Default) |
| ALPHAAj<br>OMEGAAj<br>Real and imaginary parts of Aj in radians per unit time. (Real). | Not used   | End point Aj of j-th search region in complex plane.<br>(Default = 0.0)                   | j-th shift point.<br>(Default = 0.0)   |
| ALPHABj<br>OMEGABj<br>Real and imaginary parts of Bj in radians per unit time. (Real). | Not used   | End point Bj of j-th search region in complex plane.<br>(Default = 0.0)                   | See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.           |
| Lj (Real $> 0.0$ )   | Not used   | Width of j-th search region.<br>(Default = 1.0)   | See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.           |
| NEj (Integer $> 0$ )   | Not used   | Estimated number of roots in j-th search region.  | Not used   |
| MBLKSZ<br>For block CLAN only  |  |   | Maximum Block Size<br>Default = 7  |
| IBLKSZ<br>For block CLAN only  |  |   | Initial Block Size<br>Default = 2  |

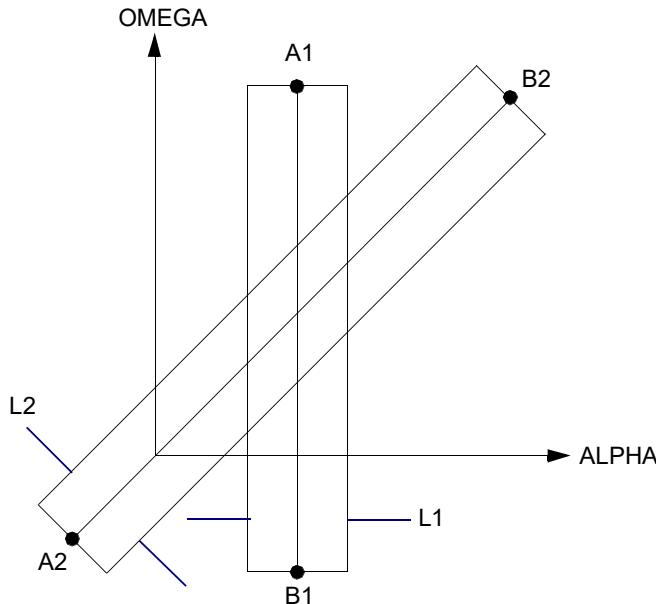


Figure 9-90      Sample Search Regions

## Remarks:

1. The EIGC entry must be selected in the Case Control Section with the command CMETHOD = SID. Methods of solution are also controlled by SYSTEM(108) on the NASTRAN statements (described in [Executing MSC Nastran, 1](#)).

| SYSTEM(108)    | Specification   |
|----------------|---|
| 0<br>(Default) | QZ HESS method or CLAN block Lanczos, as selected by EIGC entry or equivalent PARAM input. Default value. |
| 1              | Force Householder QR (Hessenberg with spill, mass matrix must be nonsingular)                             |
| 2              | Force old single vector complex Lanczos   |
| 4              | Force new block complex Lanczos   |
| 8              | Debugging output for Lanczos methods  |
| 16             | Turn off block reduction in block complex Lanczos   |
| 32             | Turn off block augmentation in block complex Lanczos  |
| 64             | Turn off full orthogonality in block complex Lanczos  |

| SYSTEM(108) | Specification  |
|-------------|--|
| 128         | Turn off preprocessing of initial vectors in block complex Lanczos     |
| 256         | Force LR method (Hessenberg, no spill, mass matrix must be invertible) |
| 512         | Force QZ method  |
| 65536       | Use semi-algebraic sort on imaginary part of roots (pre-V70.6 sort)    |

The word “force” above implies that the method selected by the system cell will be used even when another method has been selected on an EIGC entry. Sums of these values will produce two or more actions at once, when feasible. As the numbers get larger, the function is more developer-oriented than user-oriented.

- When using METHOD = “HESS”, the following should be noted:

The “HESS” method is generally more reliable and economical for small and moderate-size problems. It computes ND eigenvalues and eigenvectors.

For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, HESS may yield incorrect results. Therefore, please avoid to use HESS in linear perturbation analysis with large displacement.

- The EIGC entry may or may not require continuations as noted below.

- For the “HESS” method, continuations are not required; and their contents are ignored when present, except for ND1. However, it is recommended that continuations are not used.
- For the “CLAN” method when the continuation entry is not used a shift is calculated automatically. When a shift is input on the first continuation entry it is used as the initial shift. Only one shift is used. Data on other continuation entries is ignored.
- For METHOD = “INV”, each continuation defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.
- For all methods, if no continuation is present, then ND0 must be specified on the first entry. If a continuation is present, then NDj must be specified on the continuation and not on the first entry.

- The units of ALPHAAJ, OMEGAAJ, ALPHABJ, and OMEGABJ are radians per unit time.
- DIAG 12 prints diagnostics for the inverse power method, the complex Lanczos method and the QZ HESS method.
- If METHOD = “HESS” and the LR or QR methods (non-default methods) are selected by system cell 108 the mass matrix must be nonsingular. The default QZ method does not require a nonsingular mass matrix.
- The normalized eigenvectors may be output with the SDISPLACEMENT and/or DISPLACEMENT Case Control commands.

8. When using METHOD = CLAN, the following should be noted. The modern CLAN method (default for METHOD entry of CLAN) has been enhanced to include a block complex Lanczos approach. This method is more reliable and will not accept inaccurate roots which the old method had a tendency to do. Thus, given the same input, the new method may often accept fewer roots. For continuity the old method has been maintained and may be selected by setting SYSTEM(108). For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, CLAN is recommended.
9. The SVD method is provided for DMAP applications. If used in solution 107 or 110, and mass or damping terms are present, a user fatal exit is taken. See the MSC Web site for the Flight Loads Product examples on the use of the SVD method. The SVD operation decomposes the input stiffness matrix K into the factors U, S, and V. U and V are collections of vectors of orthogonal functions. S is a rectangular matrix with terms on the diagonal of its left partition. The factors satisfy the equation  $K = U^*S^*V'$ , where “ $^*$ ” implies complex conjugate transpose. The ND1 value has a meaning for the SVD functions which differs from eigensolution.

| ND1 | OUTPUT   |
|-----|--|
| >0  | All vectors of U and V are output.   |
| =0  | U and V are returned in a purged state.  |
| <0  | S is returned as a square matrix whose number of columns is equal to the minimum number of rows or columns of the input matrix. U and V are truncated to be commensurate with S. This is a method to reduce the costs of solving very rectangular input matrices by providing a partial solution for the most interesting vectors. |

10. For DMAP applications there are input parameters, not present in the solution sequences, that may be used to replace the function of the EIGC and CMETHOD entries. See the MSC Software Web site for details.
11. The MBLKSZ and IBKLSZ parameters are integers in concept, but must be input at real numbers (that is, with a decimal point.) They represent maximum sizes, and may be reduced internally for small size problems.
12. The IRAM method is an out-of-core implementation of the public domain ARPACK complex eigenvalue solver. To use IRAM, the mass matrix has to be symmetric or Hermitian, and the number of modes desired can be at most the order of the generalized eigenproblem minus one. IRAM does not compute the left eigenvectors and thus is not suitable for SOL200 design optimization analysis.
13. For the "IRAM" method, only the fields that contain EIGC, SID, METHOD, NORM, ND0, NDj are meaningful. The user selects the method by specifying IRAM in the METHOD field, the eigenvector normalization scheme in the NORM field, and the number of desired modes in the ND0 or the NDj field. The other fields are irrelevant.
14. If Modules are present then this entry may only be specified in the main Bulk Data section.

**EIGP****Poles in Complex Plane**

Defines poles that are used in complex eigenvalue extraction by the Determinant method.

**Format:**

| 1    | 2   | 3      | 4      | 5  | 6      | 7      | 8  | 9 | 10 |
|------|-----|--------|--------|----|--------|--------|----|---|----|
| EIGP | SID | ALPHA1 | OMEGA1 | M1 | ALPHA2 | OMEGA2 | M2 |   |    |

**Example:**

|      |    |      |     |   |     |     |   |  |  |
|------|----|------|-----|---|-----|-----|---|--|--|
| EIGP | 15 | -5.2 | 0.0 | 2 | 6.3 | 5.5 | 3 |  |  |
|------|----|------|-----|---|-----|-----|---|--|--|

| Descriptor    | Meaning   |
|---------------|---|
| SID           | Set identification number. (Integer > 0)  |
| ALPHAi,OMEGAi | Coordinates of point in complex plane. (Real)   |
| Mi            | Multiplicity of complex root at pole defined by point at ALPHAi and OMEGAi. (Integer > 0) |

**Remarks:**

1. The EIGP entry defines poles in the complex plane that are used with an associated EIGC entry having the same set number.
2. The units of ALPHAi and OMEGAi are radians per unit time.
3. Poles are used only in the determinant method. (METHOD = "DET" on the EIGC entry).
4. One or two poles may be defined on a single entry.
5. See *The NASTRAN Theoretical Manual*, Section 10.3.4, for details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**EIGR****Real Eigenvalue Extraction Data**

Defines data needed to perform real eigenvalue analysis.

**Format:**

| 1    | 2    | 3      | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|------|------|--------|----|----|----|----|---|---|----|
| EIGR | SID  | METHOD | F1 | F2 | NE | ND |   |   |    |
|      | NORM | G      | C  |    |    |    |   |   |    |

**Example:**

|      |    |     |  |  |  |    |  |  |  |
|------|----|-----|--|--|--|----|--|--|--|
| EIGR | 13 | LAN |  |  |  | 12 |  |  |  |
|------|----|-----|--|--|--|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Unique Integer > 0)   |
| METHOD     | Method of eigenvalue extraction. (Character)<br><br>Modern Methods:<br><br>LAN      Lanczos Method<br><br>AHOU     Automatic selection of HOU or MHOU method. See Remark 13.<br><br>Obsolete Methods:<br><br>INV      Inverse Power method.<br>SINV     Inverse Power method with enhancements.<br>GIV      Givens method of tridiagonalization.<br>MGIV     Modified Givens method.<br>HOU      Householder method of tridiagonalization.<br>MHOU     Modified Householder method.<br><br>AGIV     Automatic selection of METHOD = "GIV" or "MGIV". See Remark 13. |
| NORM       | Method for normalizing eigenvectors. (Character: "MASS," "MAX," or "POINT"; Default = "MASS")<br><br>MASS     Normalize to unit value of the generalized mass. (Default)<br>MAX      Normalize to unit value of the largest component in the analysis set.<br>POINT     Normalize to a positive or negative unit value of the component defined in fields 3 and 4. The POINT option is not supported for METHOD=LAN. (Defaults to "MASS" if defined component is zero.)   |

| Descriptor | Meaning  |
|------------|--|
| G          | Grid or scalar point identification number. Required only if NORM = "POINT".<br>(Integer > 0)        |
| C          | Component number. Required only if NORM = "POINT" and G is a geometric grid point. (1 ≤ Integer ≤ 6) |

Table 9-12      Relationship Between METHOD Field and Other Fields for Obsolete Methods

| Field  | METHOD Field  |   |
|--------|---|---|
|        | INV or SINV   | GIV, MGIV, HOU, or MHOU   |
| F1, F2 | Frequency range of interest. F1 must be input. If METHOD = "SINV" and ND, is blank, then F2 must be input. See also Remark 21.<br>(Real ≥ 0.0)  | Frequency range of interest. If ND is not blank, F1 and F2 are ignored. If ND is blank, eigenvectors are found with natural frequencies that lie in the range between F1 and F2. (Real ≥ 0.0; F1 < F2)  |
| NE     | Estimate of number of roots in range (Required for METHOD = "INV"). Not used by "SINV" method.<br>(Integer > 0)   | Not used.   |
| ND     | Desired number of roots. If this field is blank and METHOD = "SINV", then all roots between F1 and F2 are searched and the limit is 600 roots.<br>(Integer > 0, Default is $3 \cdot NE$ for METHOD = "INV" only.) | Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPPORTi entries. (Integer ≥ 0; Default = 0) |

#### Remarks:

1. The EIGR entry must be selected with the Case Control command METHOD = SID.
2. See [Real Eigenvalue Analysis](#) in the *MSC Nastran Dynamic Analysis Guide* for a discussion of method selection.
3. The units of F1 and F2 are cycles per unit time.
4. The continuation entry is optional. If the continuation entry is not specified, then mass normalization is performed.
5. The contemporary methods are LAN and AHOU. The other methods are in a maintenance-only status, with no enhancements planned for them. They may be eliminated in a future release of Nastran.

6. The LAN method is the most general-purpose method, and may be used on both small- and large-size problems. It takes advantage of sparsity of input matrices, leading to greater efficiency on large-size problems. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry. The NE, G, and C fields are ignored for the LAN method. The NORM field may be set to MASS (the default value) or MAX. The conventions used when both the Fi and ND fields are specified are described in [Table 9-13](#) of the EIGRL entry description. The EIGRL entry is an alternate method to select the LAN method. It has several other input options for special cases. When both EIGRL and EIGR have the same SID and that SID is selected by a METHOD command the EIGRL entry takes precedence.
7. The AHOU method is competitive with the LAN method when there are small, dense matrices and many eigenvectors are required. This most commonly occurs when static or dynamic reduction is performed. The AHOU method does not take advantage of matrix sparsity, so that computation cost rises with the cube of the number of DOFs. The AHOU method responds to all permitted values for all the other fields except NE, which is ignored.
8. All methods require a positive semi-definite (psd) mass matrix for stable solutions. The mass matrix may be tested for this condition for all methods of solution by setting SYSTEM(303). A value of “-4” should be sufficient to identify problem matrices. A fatal error exit is taken when it is not met. All Nastran metric elements are designed to produce psd mass matrices. CMASSi elements, DMIG matrices selected by the M2GG command, and matrices input via INPUTT4 are special methods that allow addition of non-psd terms by use of non-metric element input. If none of this type of special input is present and the fatal error exit is taken you may have encountered an error in a metric element. Contact your local MSC technical representative for corrective action in this case.
9. The LAN and AHOU methods allow singular but positive semi-definite mass matrices.
10. The tridiagonal methods include the xGIV and xHOU methods, where “x” is described in the following comments. All tridiagonal methods compute all eigenvalues, and the number of eigenvectors specified by the Fi and Nd fields, as described in [Table 9-12](#).
11. If “x” is blank (for example, the HOU method is selected) the mass matrix must be non-singular.
12. If “x” is M (for example, the MHOU method is selected) the mass matrix may be singular. A modified, shifted problem is solved in an inverse basis with this method. Some precision in the solution and longer computation time is exchanged for a more stable solution.
13. If “x” is A (for example, the AHOU method is selected) an attempt is made to solve the problem without shifting, in the interest of cost reduction and maximum precision. If the mass matrix is determined to be poorly conditioned for inversion the basis is automatically shifted with the modified method.
14. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
15. If NORM = “POINT”, the selected component should be in the analysis set (a-set). (The program uses NORM = “MAX” when it is not in the analysis set.) The displacement value at the selected component will be positive or negative unity.

16. The “SINV” method is an enhanced version of the “INV” method. It uses Sturm sequence number techniques to make it more likely that all roots in the range have been found. It is generally more reliable and more efficient than the “INV” method.
17. For the “INV” and “SINV” methods, convergence is achieved at  $10^{-6}$ . Convergence is tested by other criteria for the other methods.
18. For the “SINV” method only, if F2 is blank, the first shift will be made at F1, and only one eigensolution above F1 will be calculated. If there are no modes below F1, it is likely that the first mode will be calculated. If there are modes below F1 (including rigid body modes defined by SUPORT entries), a mode higher than the first mode above F1 may be calculated.
19. When F1, F2, and ND are all zero or blank, ND is reset to 1. A User Warning Message is produced for this condition, which is interpreted as likely to be due to an inadvertent omission by the user.
20. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the F1 and F2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
21. F2 must be specified if DOMAIN SOLVER ACMS or DOMAIN SOLVER MODES is also specified in the Executive Control Section.
22. For large sized problems, Lanczos is the most efficient and is the recommended method for large problems.
23. It is not recommended to use AHOU, HOU, MHOU if RIGID=LAGRAN case control.
24. If Modules are present then this entry may only be specified in the main Bulk Data section.

**EIGRL**

## Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

**Format:**

| 1   | 2   | 3  | 4  | 5  | 6       | 7      | 8      | 9    | 10 |
|---|-----|----|----|----|---------|--------|--------|------|----|
| EIGRL   | SID | V1 | V2 | ND | MSG_LVL | MAXSET | SHFSCL | NORM |    |
| option_1 = value_1    option_2 = value_2,    etc. |     |    |    |    |         |        |        |      |    |

**Example:**

|                    |   |     |     |    |  |  |  |  |  |
|--------------------|---|-----|-----|----|--|--|--|--|--|
| EIGRL              | 1 | 0.1 | 3.2 | 10 |  |  |  |  |  |
| NORM=MAX    NUMS=2 |   |     |     |    |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Unique Integer > 0)   |
| V1, V2     | For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remark 4. (Real or blank, $-5 \times 10^{16} \leq V1 < V2 \leq 5. \times 10^{16}$ )   |
| ND         | Number of roots desired. See Remark 4. (Integer > 0 or blank)   |
| MSG_LVL    | Diagnostic level. ( $0 \leq \text{Integer} \leq 4$ ; Default = 0)   |
| MAXSET     | Number of vectors in block or set. Default is machine dependent. See Remark 14.   |
| SHFSCL     | Estimate of the first flexible mode natural frequency. See Remark 10. (Real or blank)   |
| NORM       | Method for normalizing eigenvectors (Character: "MASS" or "MAX") <ul style="list-style-type: none"> <li>MASS      Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)</li> <li>MAX      Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)</li> </ul> |
| ALPH       | Specifies a constant for the calculation of frequencies ( $F_i$ ) at the upper boundary segments for the parallel method based on the following formula. See Remark 13. (Real > 0.0; Default = 1.0): $\text{ALPH} < 1.0, F_i = V1 + (V2 - V1) \frac{1 - \text{ALPH}^i}{1 - \text{ALPH}^{\text{NUMS}}}$ $\text{ALPH} = 1.0, F_i = V1 + (V2 - V1) \frac{i}{\text{NUMS}}$  |
| NUMS       | Number of frequency segments for the parallel method. (Integer > 0; Default = 1)  |

| Descriptor        | Meaning  |
|-------------------|--|
| Fi                | Frequency at the upper boundary of the i-th segment. See Remark 13. (Real or blank; $V1 < F1 < F2 < \dots < F15 < V2$ )  |
| option_i= value_i | Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGLVL, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry. |

**Remarks:**

1. Real eigenvalue extraction data sets must be selected with the Case Control command METHOD = SID.
2. The units of V1 and V2 are cycles per unit time in vibration analysis, and are eigenvalues in buckling analysis. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.
3. NORM = "MASS" is ignored in buckling analysis and NORM = "MAX" will be applied.
4. The roots are found in order of increasing magnitude; that is, those closest to zero are found first. The number and type of roots to be found can be determined from [Table 9-13](#).

Table 9-13 Number and Type of Roots Found with EIGRL Entry

| V1    | V2    | ND    | Number and Type of Roots Found                   |
|-------|-------|-------|--|
| V1    | V2    | ND    | Lowest ND or all in range, whichever is smaller. |
| V1    | V2    | blank | All in range                                     |
| V1    | blank | ND    | Lowest ND in range $[V1, +\infty]$               |
| V1    | blank | blank | Lowest root in range $[V1, +\infty]$             |
| blank | blank | ND    | Lowest ND roots in $[-\infty, +\infty]$          |
| blank | blank | blank | Lowest root. See Remark 11.                      |
| blank | V2    | ND    | Lowest ND roots below V2                         |
| blank | V2    | blank | All below V2                                     |

5. In vibration analysis, if  $V1 < 0.0$ , the negative eigenvalue range will be searched. (Eigenvalues are proportional to  $V_i$  squared; therefore, the negative sign would be lost.) This is a means for diagnosing improbable models. In buckling analysis, negative V1 and/or V2 require no special logic.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.

8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. It may also be reset if there is insufficient memory available. The default value is recommended.
9. In vibration analysis, if V1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If V1 is set to zero, negative eigenvalues are not calculated.
10. A specification for SHFSCL may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. However, this method will not output the additional roots.
12. NASTRAN SYSTEM(146) (also known as FBSMEM) provides options for I/O reduction during FBS operations according to the following table.

**Table 16      SYSTEM(146) Options**

| SYSTEM(146)<br>(default=0) | Description   |
|----------------------------|---|
| -1 or 0                    | Use available memory to store as much of the factor matrix as possible during FBS operations. System Information Message 4199 is printed in the F04 file. |
| 1                          | Disable all special memory operations for FBS.  |
| 2                          | Increase memory reserved for sparse method by approximately 200%.   |
| 3                          | Increase memory reserved for sparse method by approximately 300%.   |
| 4                          | Increase memory reserved for sparse method by approximately 400%.   |

13. For the distributed parallel method, the frequency range between V1 and V2 may be subdivided into segments that can then be analyzed in parallel. V1 and V2 must be specified for the parallel method. NUMS must be specified greater than 1 to take advantage of the parallel method. NUMS may also be specified on the NUMSEG keyword of the NASTRAN statement. Currently, NUMSEG must equal the number of processors and by default NUMSEG is set to the number of processors requested by the DMP keyword. If both are specified, then NUMS takes precedence.  
The upper frequencies of each segment may be generated automatically by ALPH or specified directly in Fi. If both are specified, then Fi takes precedence over ALPH as long as they are consistent. ALPH if multiplied by 100 may also be specified on FRQSEQ keyword of the NASTRAN statement.
14. Increasing MAXSET may improve performance for large problems where a large number of eigenvalues are being found. The default is 7 on all machines. SYSTEM(263) may be set in an rcfle to effectively modify the default; however the setting on the EIGRL entry always takes precedence.

15. SYSTEM(196), keyword SCRSAVE, controls reuse of scratch files when segment logic is invoked. SYSTEM(196) is useful only when multiple frequency segments are requested on a Lanczos run. (Multiple frequency segments can be requested via the NUMS field in the EIGRL entry and by SYSTEM(197).) Each frequency segment requires a minimum of three scratch files. When multiple frequency segments are used on a single processor computer then each frequency segment is solved serially. In this case, it makes sense to let segment #2 use the scratch files which were used by segment #1 since work for segment #1 has been completed (otherwise it wouldn't be working on #2). Similarly, when work for segment #2 is finished, segment #3 should be able to use #2's scratch files. SYSTEM(196)=1 allows such file reuse and is considered a safe default on Version 70 and later systems.
16. The new buckling shift logic in Version 70.5 tends to shift to 1.0 first. The logic may have difficulty finding the lowest ND roots if a problem requests a small number of roots (ND) when there are thousands of roots below 1. In this case either the loading should be scaled, SHFSCL specified, or a smaller frequency range requested.
17. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry.
18. V2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.
19. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the V1 and V2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
20. For buckling solutions, if V1 is explicitly set to 0.0, negative eigenvalues will be calculated based on a negative shift value. This is done to avoid singularities resulting from a shift at 0.0. If V1 is set to a small positive value, then negative eigenvalues will not be calculated. This behavior was changed with the MSC Nastran 2010 release whereas before setting V1 to 0.0 yielded no negative eigenvalues. The best approach, however, is to follow the suggestion from Remark 19.
21. If Modules are present then this entry may only be specified in the main Bulk Data section.

**ELEMUDS**

## Element Property User Defined Service or Subroutine

Allows the user to provide element property routines for use with specified Nastran elements.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| ELEMUDS | PID    | PTYPE  | GROUP  | UNAME  | DEPEN  | NMLSPD |        |        |    |
|         | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5 | IDATA6 | IDATA7 |    |
|         |        | IDATA8 | IDATA9 | ...    | ...    | IDATAn |        |        |    |
|         | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5 | RDATA6 | RDATA7 |    |
|         |        | RDATA8 | RDATA9 | ...    | ...    | RDATAn |        |        |    |
|         | “CHAR” | CDATA1 | CDATA2 | ...    | ...    | CDATAn |        |        |    |

**Examples:**

*In FMS Section of Nastran input stream:*

```
CONNECT SERVICE ELEMENT 'SCA.MDSolver.Util.Ums'
```

*In Bulk Data:*

|         |    |         |         |       |      |  |  |  |  |
|---------|----|---------|---------|-------|------|--|--|--|--|
| ELEMUDS | 17 | PBUSH2D | ELEMENT | THPAD | FREQ |  |  |  |  |
|---------|----|---------|---------|-------|------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Element property identification number that matches the identification number on a PBUSH2D element property entry. (Integer > 0)  |
| PTYPE      | The name of the element entry. Currently supported element property entry is: PBUSH2D. (Character; no Default)  |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)  |
| UNAME      | User subroutine name associated with the entry. See Remark 6.(Character)  |
| DEPEN      | Specifies if element is frequency dependent. DEPEN=NOM or blank; user element is not frequency dependent. DEPEN=FREQ; user element is frequency dependent. (Character default=NOM)  |
| NMLSPD     | Nominal speed (cycles per second) used by the CBUSH2DA to calculate nominal stiffness, mass and damping terms. Nominal terms will be used during reduction and real eigenvalue solution. In a frequency-dependent problem, updated values will be calculated at each frequency. |
| “INT”      | Keyword indicating that the following data is integer. (Character)  |
| IDATAi     | Additional user supplied Integer data not already existing on the specified property entry. (Integer; no Default)   |
| “REAL”     | Keyword indicating that the following data is real. (Character)   |

|         |   |
|---------|---|
| RDATAi  | Additional user supplied Real data not already existing on the specified property entry.<br>(Real; no Default)        |
| “CHAR”  | Keyword indicating that the following data is Character. (Character)  |
| CDATAAi | Additional user supplied Character data not already existing on the specified property entry. (Character; no Default) |

**Remarks:**

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing PID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
5. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. If the ROMAC interface is being used then UNAME=THPAD is required. See the THPAD Bulk Data entry (p. 3244) for a description of ROMAC service. (<http://www.virginia.edu/romac/>)

**ELEMUDS**

## Element Property User Defined Service or Subroutine - SOL 600

Allows the user to provide element property routines for use with enhanced nonlinear elements in SOL 600.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| ELEMUDS | PID    | PTYPE  | GROUP  | UNAME  |        |        |        |        |    |
|         | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5 | IDATA6 | IDATA7 |    |
|         |        | IDATA8 | IDATA9 | ...    | ...    | IDATAn |        |        |    |
|         | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5 | RDATA6 | RDATA7 |    |
|         |        | RDATA8 | RDATA9 | ...    | ...    | RDATAn |        |        |    |
|         | “CHAR” | CDATA1 | CDATA2 | ...    | ...    | CDATAn |        |        |    |

**Examples:**

|         |      |        |         |      |  |  |  |  |  |
|---------|------|--------|---------|------|--|--|--|--|--|
| ELEMUDS | 17   | PSHELL | UELEM   | RTN1 |  |  |  |  |  |
| ELEMUDS | 17   | PSHELL | UELEM1  | RTN3 |  |  |  |  |  |
|         | REAL | .00134 | 1.467+4 | .03  |  |  |  |  |  |
|         | INT  | 8      | 3       |      |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Element property identification number that matches the identification number on a PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD element property entry. (Integer > 0) |
| PTYPE      | The name of the element entry. Currently supported element property entries include: PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD. (Character; no Default)            |
| GROUP      | The group name used for the FMS section CONNECT SERVICE entry. (Character; no Default)   |
| UNAME      | A primary or secondary name to identify the user subroutine. See Remark 6. (Character; Default blank)  |
| “INT”      | Keyword indicating that the following data is integer. (Character)   |
| IDATAi     | Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)   |
| “REAL”     | Keyword indicating that the following data is real. (Character)  |
| RDATAi     | Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)   |

|         |   |
|---------|---|
| "CHAR"  | Keyword indicating that the following data is Character. See Remark 7. (Character)                                  |
| CDATAAi | Additional user supplied Character data not already existing on the specified MAT entry.<br>(Character; no Default) |

**Remarks:**

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials.
2. PID must match an existing PID.
3. A CDATAi entry cannot be the Character "REAL", "INT", or "CHAR".
4. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
5. The ELEMUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character\*16 name in common block /userch/.
6. UNAME identifies the user subroutine name to be called.
7. Character fields CDATAi are ignored in SOL 600 are not passed to the user subroutine.

**ELIST****Element List**

Defines a list of CQUAD4 and CTRIA3 structural elements for virtual fluid mass.

**Format:**

| 1     | 2   | 3  | 4   | 5      | 6  | 7  | 8  | 9  | 10 |
|-------|-----|----|-----|--------|----|----|----|----|----|
| ELIST | LID | E1 | E2  | E3     | E4 | E5 | E6 | E7 |    |
|       | E8  | E9 | E10 | -etc.- |    |    |    |    |    |

**Example:**

|       |   |    |     |    |      |     |     |  |  |
|-------|---|----|-----|----|------|-----|-----|--|--|
| ELIST | 3 | 51 | -62 | 68 | THRU | 102 | 122 |  |  |
|-------|---|----|-----|----|------|-----|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| LID        | Identification number of list. (Integer > 0)   |
| Ei         | Identification number of a structural element. See Remark 1. for the meaning of the negative sign. The string “THRU” may be used to indicate that all existing elements between those referenced in the preceding and succeeding fields are in the list. (Integer ≠ 0 or “THRU”) |

**Remarks:**

1. If the ELIST entry is referenced by field 6 of an MFLUID entry, the wetted side of the element is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal as determined by applying the right-hand rule to the sequence of its corner points. If the “THRU” option is used, then immediately preceding and succeeding elements must have the same sign.
2. Large open “THRUs” should be avoided.
3. The word “THRUs” must not appear in field 2 or 9 on the parent entry or on any continuations.
4. If any ELIST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, ELIST entry changes or additions are not recommended on restart.

## **ENDDATA**

Bulk Data Delimiter

Designates the end of the Bulk Data Section.

Format:

ENDDATA

Remark:

1. ENDDATA is optional.

**ENDDYNA**

Defines the End of Direct Text to Dytran

All entries between TODYNA and ENDDYNA will be passed directly by SOL 700 to Dytran. Used in SOL 700 only.

Format:

| 1       | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|---|---|---|---|---|---|---|---|----|
| ENDDYNA |   |   |   |   |   |   |   |   |    |

Example:

|         |     |        |       |        |  |  |  |  |  |
|---------|-----|--------|-------|--------|--|--|--|--|--|
| TODYNA  |     |        |       |        |  |  |  |  |  |
| MAT1    | 345 | 29.0E6 | 0.285 | 0.0004 |  |  |  |  |  |
| ENDDYNA |     |        |       |        |  |  |  |  |  |

| Descriptor | Meaning |
|------------|---------|
| TODYNA     |         |
| MAT1       |         |
| ENDDYNA    |         |

Remarks:

See [TODYNA, 3272](#) for details of how this entry is used.

## ENTUDS

### User Defined Logic at Entry Point

Calls user defined logic within a SCA service at the point specified within the solution sequence.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7                  | 8      | 9      | 10 |
|--------|--------|--------|--------|--------|--------|--------------------|--------|--------|----|
| ENTUDS | ENTID  | ENTPNT | GROUP  |        |        |                    |        |        |    |
|        | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5             | IDATA6 | IDATA7 |    |
|        |        | IDATA8 | IDATA9 | ...    | ...    | IDATA <sub>n</sub> |        |        |    |
|        | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5             | RDATA6 | RDATA7 |    |
|        |        | RDATA8 | RDATA9 | ...    | ...    | RDATA <sub>n</sub> |        |        |    |
|        | “CHAR” | CDATA1 | CDATA2 | ...    | ...    | CDATA <sub>n</sub> |        |        |    |

**Example:**

|        |      |        |         |  |  |  |  |  |  |
|--------|------|--------|---------|--|--|--|--|--|--|
| ENTUDS | 1    | RCENT1 | MY_FUNC |  |  |  |  |  |  |
|        | INT  | 2      | 17      |  |  |  |  |  |  |
|        | REAL | .5     | .25     |  |  |  |  |  |  |

| Descriptor         | Meaning   |
|--------------------|---|
| ENTID              | Entry point identification number (Integer > 0)   |
| ENTPNT             | The point of entry in the solution sequence. Acceptable values for now: “RCENT1”, “RCENT2”, “RCOUT1”, “RCEEXEC1”. (Character; Required) |
| GROUP              | The SCA group name used to identify the service (Character; Required)   |
| “INT”              | Key word indicating that the following data is integer. (Character)   |
| IDATA <sub>i</sub> | Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)                            |
| “REAL”             | Key word indicating that the following data is real. (Character)  |
| RDATA <sub>i</sub> | Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)                                  |
| “CHAR”             | Key word indicating that the following data is Character. (Character)   |
| CDATA <sub>i</sub> | Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)                        |

**Remarks:**

1. This entry is for RC Network solver only.
2. In SINDA input file, there are 4 entry points for users to input customized logics.
  - a. RCENT1 ---- Variable 1 block

- b. RCENT2 ---- Variable 2 block
- c. RCOUT1 ---- Output block
- d. RCEEXEC1 ---- Execution block

**EOSDEF**

Deflagration - SOL 700 only

EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas.

**Format**

| 1      | 2   | 3     | 4     | 5   | 6              | 7              | 8    | 9    | 10       |
|--------|-----|-------|-------|-----|----------------|----------------|------|------|----------|
| EOSDEF | MID | GAMMA | B     | R   | C <sub>v</sub> | C <sub>p</sub> | E    | RHOS | +EOSDEF1 |
| EOSDEF | 3   | 1.123 | 0.001 | 304 |                |                | 1E+6 | 1600 | +EOSDEF1 |

**Example**

| 1        | 2    | 3    | 4    | 5    | 6   | 7   | 8 | 9 | 10 |
|----------|------|------|------|------|-----|-----|---|---|----|
| +EOSDEF1 | RHOF | W    | BETA | SAVR | X   | Y   |   |   |    |
| +EOSDEF1 | 0.1  | 4E-8 | 0.85 | 6000 | 0.6 | 0.0 |   |   |    |

| Field | Contents  | Type   | Default            |
|-------|---|--------|--------------------|
| MID   | Unique material number.                                     | I > 0  | Required           |
| GAMMA | Constant $\gamma$   | R>0    | Required           |
| B     | Gas co-volume   | R≥ 0   | Required           |
| R     | Gas constant.   | R>0    | Refer to remark 2. |
| CV    | Specific heat at constant volume                            | R>0    | Refer to remark 3. |
| CP    | Specific heat at constant pressure                          | R>0    | Refer to remark 3. |
| E     | Chemical energy per unit burned mass                        | R>0.0  | Required           |
| RHOS  | Reference density of propellant                             | R>0    | Required           |
| RHOF  | <u>Density of powder</u><br>reference density of propellant | R>0    | Refer to remark 5. |
| W     | Burning rate coefficient                                    | R≥ 0.0 | Required           |
| BETA  | Burning rate exponent                                       | R≥ 0.0 | Required           |
| SAVR  | Initial surface area divided by volume                      | R>0    | Required           |
| X     | Parameter form function                                     | R≥ 0.0 | Required           |
| Y     | Parameter form function                                     | R≥ 0.0 | Required           |

## Remarks

1. EOSDEF – The deflagration Equation of State is explained in *Dytran Theory Manual*, Chapter 4: Models.
2. This equation of state is used with Eulerian elements.
3. The temperature of gas is calculated when one of the constants R, C<sub>v</sub> or C<sub>p</sub> are specified. When temperature is not mentioned in an output request, omit the he constants .
4. The pressure in the reaction products is defined by the Noble-Abel equation of state as follows:

$$\rho = (\gamma - 1) \frac{\rho}{1 - b\rho} e \text{ for reacted product,}$$

$$T = \frac{(\gamma - 1)e}{R} \text{ where } \gamma, b \text{ are constants and } R \text{ is the gas constant.}$$

The chemical reaction rate for conversion of un-reacted explosive to reaction products is mentioned the relation below:

$$\xi = w SAVR$$

vivacity

$$\phi = (1 - F)^X + YF$$

form function

$$\frac{dF}{dt} = \xi \phi p^\beta$$

time derivative of burn fraction

where;

w = burning rate coefficient

$\beta$  = burning rate exponent

SAVR = initial surface area divided by volume

Y= parameter form function

S= parameter form function

5. RHOF allows taking into account the air between gunpowder grains. Also, it allows taking into account air inside the combustion chamber. For example If the chamber is filled with 10% real propellant and the rest is filled with air then RHOF = 0.1.
6. To indicate what Eulerian regions can burn, the variable DEFMAT can be used. In regions that can burn DEFMAT=1. DEFMAT can be set on the TICVAL card:  
TICVAL,14,,DENSITY,1.14,SIE,300000.2,DEFMAT,1.0
7. Ignition of the propellant approached by setting SIE to the flame temperature in specific areas:  
TICVAL,15,DENSITY,1.14,SIE,4231908.591,DEFMAT,1.0  
Where SIE<sub>ignition</sub> = Flame Temp x Cv  
Flame Temp can be extracted from Chemical Energy per unit burned mass and Gas constant: T<sub>flame</sub> = E / R .

**EOSGAM****Gamma Law Gas Equation of State - SOL 700 only**

Defines the properties of a Gamma Law equation of state where the pressure p is defined as:

$$p = (\gamma - 1)\rho e$$

where:

$e$  = specific internal energy per unit mass

$\rho$  = overall material density

$\gamma$  = A constant

Used in SOL 700 only.

Format:

| 1      | 2   | 3     | 4 | 5  | 6  | 7    | 8 | 9 | 10 |
|--------|-----|-------|---|----|----|------|---|---|----|
| EOSGAM | EID | GAMMA | R | CV | CP | VISC |   |   |    |

Example:

|        |   |     |       |  |  |  |  |  |  |
|--------|---|-----|-------|--|--|--|--|--|--|
| EOSGAM | 2 | 1.4 | 287.0 |  |  |  |  |  |  |
|--------|---|-----|-------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique equation of state number. (Integer > 0, Required)   |
| GAMMA      | Constant $\gamma$ . (Real $\geq 0$ , default=blank)  |
| R          | Gas constant. See Remarks 1. and 3. (Real > 0, default=blank)  |
| CV         | Specific heat at constant volume. See Remarks 1. and 3. (Real > 0, default=blank)                        |
| CP         | Specific heat at constant pressure. See Remarks 1. and 3. (Real > 0, default=blank)                      |
| VISC       | Viscosity coefficient. If zero, no viscosity. See Remarks 4., 5., and 6. (Real $\geq 0$ , default=blank) |

Remarks:

1. The temperature of the gas will be calculated when one of the gas constants,  $R$ ,  $C_v$  or  $C_p$  is specified.
2. The Euler variable name for temperature is TEMPTURE.
3.  $\gamma$ ,  $R$ ,  $C_v$  and  $C_p$  have the following relationships:

$$\gamma = \frac{C_p}{C_v} \quad R = C_p - C_v$$

4. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient. The SI-unit of viscosity is

$$\text{Pa} \cdot \text{s} = \frac{\text{N}_\text{s}}{\text{m}^2} = \frac{\text{kg}}{\text{ms}}$$

5. If possible, use (in coupled analysis) the FASTCOUP coupling algorithm because viscous fluxes are computed more accurately for fast coupling than for general coupling.
6. For the single mat solver, viscous stresses can be requested by the use of TXX through TZx. Also, EFFSTS is available. For the multi-material solver, viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZx-VIS. This viscous stresses depend only on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stress. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.

**EOSIG****Ignition and Growth Equation of State - SOL 700 only**

Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3       | 4      | 5       | 6      | 7    | 8 | 9 | 10 |
|-------|-----|---------|--------|---------|--------|------|---|---|----|
| EOSIG | EID | UNITDEF | DBEXPL | UNITCNV |        |      |   |   | +  |
| +     |     | AE      | BE     | R1E     | R2E    | OMGE | I | G | +  |
| +     | A   | AP      | BP     | R1P     | R2P    | OMGP | X | Y | +  |
| +     | Z   | R       | ECHEM  | PRSTOL  | ITRMAX |      |   |   |    |

**Example:**

|       |    |  |      |    |  |  |  |  |  |
|-------|----|--|------|----|--|--|--|--|--|
| EOSIG | 10 |  | LX17 | SI |  |  |  |  |  |
|-------|----|--|------|----|--|--|--|--|--|

| Descriptor | Meaning  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
|------------|--|-------|---|------|----------------------------|--------|----------------|-------|----------------|------|----------------------|--------|-----------------------|--------|---------------------------|------|------------------------|------|-----------------|
| EID        | Unique equation of state number referenced from MATDEOL. (Integer > 0; Required)   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| UNITDEF    | User-defined default unit for the inputs: See Remarks 2. and 3. (Character) <table> <tr> <td>CCGMS</td><td>m/g/<math>\mu</math>s units</td></tr> <tr> <td>SI</td><td>International System units</td></tr> <tr> <td>METRIC</td><td>Metric units</td></tr> <tr> <td>IMPER</td><td>imperial units</td></tr> <tr> <td>MMMG</td><td>mm/mg/<math>\mu</math>s units</td></tr> </table>  | CCGMS | m/g/ $\mu$ s units                      | SI   | International System units | METRIC | Metric units   | IMPER | imperial units | MMMG | mm/mg/ $\mu$ s units |        |                       |        |                           |      |                        |      |                 |
| CCGMS      | m/g/ $\mu$ s units   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| SI         | International System units   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| METRIC     | Metric units   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| IMPER      | imperial units   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| MMMG       | mm/mg/ $\mu$ s units   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| DBEXPL     | Use explosive material from the database (See Remarks 4. and 6.) (Character, NO). The following detonation materials are available in the data base: <table> <tr> <td>NO</td><td>The database is not used. See Remark 5.</td></tr> <tr> <td>P94A</td><td>PBX-9404 (a) explosive</td></tr> <tr> <td>TATB</td><td>TATB explosive</td></tr> <tr> <td>PENT</td><td>PETN explosive</td></tr> <tr> <td>CTNT</td><td>Cast TNT explosive</td></tr> <tr> <td>LCOMPB</td><td>LANL COMP B explosive</td></tr> <tr> <td>MCOMPB</td><td>Military COMP B explosive</td></tr> <tr> <td>P94B</td><td>PBX-9404 (b) explosive</td></tr> <tr> <td>LX17</td><td>LX-17 explosive</td></tr> </table> | NO    | The database is not used. See Remark 5. | P94A | PBX-9404 (a) explosive     | TATB   | TATB explosive | PENT  | PETN explosive | CTNT | Cast TNT explosive   | LCOMPB | LANL COMP B explosive | MCOMPB | Military COMP B explosive | P94B | PBX-9404 (b) explosive | LX17 | LX-17 explosive |
| NO         | The database is not used. See Remark 5.  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| P94A       | PBX-9404 (a) explosive   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| TATB       | TATB explosive   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| PENT       | PETN explosive   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| CTNT       | Cast TNT explosive   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| LCOMPB     | LANL COMP B explosive  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| MCOMPB     | Military COMP B explosive  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| P94B       | PBX-9404 (b) explosive   |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| LX17       | LX-17 explosive  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| UNITCNV    | User defined conversion units: (Character; see Remarks 2. and 3.) <table> <tr> <td>CGMS</td><td>cm/g/<math>\mu</math>s units</td></tr> </table>  | CGMS  | cm/g/ $\mu$ s units                     |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |
| CGMS       | cm/g/ $\mu$ s units  |       |   |      |                            |        |                |       |                |      |                      |        |                       |        |                           |      |                        |      |                 |

| Descriptor | Meaning  |
|------------|--|
| SI         | International System units   |
| METRIC     | Metric units   |
| IMPER      | Imperial units   |
| MMMGSS     | mm/mg/ $\mu$ s units   |
| AE         | Constant $A_e$ for un-reacted explosive. See Remark 5. (Real; Required)                  |
| BE         | Constant $B_e$ for un-reacted explosive. See Remark 5. (Real; Required)                  |
| R1E        | Constant $R_{1e}$ for un-reacted explosive. See Remark 5. (Real; Required)               |
| R2E        | Constant $R_{2e}$ for un-reacted explosive. See Remark 5. (Real; Required)               |
| OMGE       | Constant $\omega_e$ for un-reacted explosive   |
| I          | First ignition coefficient. See Remark 5. (Real; Required)                               |
| G          | Second ignition coefficient. See Remark 5. (Real; Required)                              |
| A          | Density ignition coefficient. See Remark 5. (Real; Required)                             |
| AP         | Constant $A_p$ for reacted product. See Remark 5. (Real; Required)                       |
| BP         | Constant $B_p$ for reacted product. See Remark 5. (Real; Required)                       |
| R1P        | Constant $R_{1p}$ for reacted product. See Remark 5. (Real; Required)                    |
| R2P        | Constant $R_{2p}$ for reacted product. See Remark 5. (Real; Required)                    |
| OMGP       | Constant $\omega_p$ for reacted product. See Remark 5. (Real; Required)                  |
| X          | Surface burning exponent. See Remark 5. (Real, 2./9.)                                    |
| Y          | Surface burning exponent. See Remark 5. (Real, 2./3.)                                    |
| Z          | Pressure exponent. See Remark 5. (Real; Required)  |
| R          | Relative density exponent. See Remark 5. (Real, .4)                                      |
| ECHEM      | Chemical energy of high explosive per unit mass. See Remark 5. (Real; Required)          |
| PRSTOL     | Tolerance for pressure equilibrium iterations in mixed phase elements. (Real > 0, 1.E-6) |
| ITRMAX     | Maximum number of iterations in pressure equilibrium iterations. (Integer > 0, 16)       |

### Remarks:

1. This equation of state can only be used with solid Eulerian elements.
2. The definition of the unit system in which the input values are defined is required information only in case you wish to have an automatic conversion to a different unit system as defined by the UNITCNV field. In case you are using the conversion mechanism, note that the density RHO in the corresponding MATDEUL entry will be interpreted in the unit system defined here. [Table 9-14](#) defines sets of units available:

Table 9-14 Sets of Units Used in the IG Model

| Quantity    | CGμs                              | SI                | Metric            | Imperial                            | MMMGμs             |
|-------------|-----------------------------------|-------------------|-------------------|-------------------------------------|--------------------|
| Length      | Centimeter (cm)                   | Meter (m)         | Centimeter (cm)   | Inch (in)                           | Millimeter (mm)    |
| Time        | Microsecond (μs)                  | Second (s)        | Second (s)        | Second (s)                          | Microsecond (μs)   |
| Mass        | Gram (g)                          | Kilogram (kg)     | Gram (g)          | Slug (lbf-s <sup>2</sup> /in)       | Milligram (mg)     |
| Force       | Teradyne                          | Newton (N)        | Dyne              | Pound force (lbf)                   | kN                 |
| Density     | g/cm <sup>3</sup>                 | Kg/m <sup>3</sup> | g/cm <sup>3</sup> | lbf-s <sup>2</sup> /in <sup>4</sup> | mg/mm <sup>3</sup> |
| Stress      | Mbar                              | Pascal (Pa)       | μbar              | Lbf/in <sup>2</sup>                 | GPa                |
| Energy      | 1012 erg (Mbars-cm <sup>3</sup> ) | Joule (J)         | Erg               | Lbf-in                              | J                  |
| Temperature | Kelvin (K)                        | Kelvin (K)        | Kelvin (K)        | Kelvin (K)                          | Kelvin (K)         |

- The UNITCNV field defines the unit system to which the material parameters are converted. In case you are not using one of the database material models, you also have to define the default unit system (UNITDEF) in which you supplied the data.
- You can use the database containing several detonation materials to start the analysis. The material data are taken from Lee/Tarver (Ref. 1) and Murphy/Lee (Ref. 2) papers in the Theory Manual. The equations of state parameters are given in the [Table 9-15](#).
- The default setting for DBEXPL is NO, which means you should define the values in the input fields (fields 12 to 36). If the database material name is defined, all values in the input fields will be overridden. The reference density RHO defined on the corresponding MATDEUL entry will be set to the value from the database.
- The default unit system for the material database parameters is the CGMS unit system. If you wish to use the material base data in a different unit system, you can specify this by defining the target unit system in the UNITCNV field.
- You can define the shear property and yield model of the material with respectively SHXXX and YLDXX entry. Note that the unit system of data required in these entries should be consistent with the unit system defined in the UNITCNV field.
- The IG equation of state cannot be used in combination with a spallation model.
- The following JWL equation of state is used to calculate the pressure of the un-reacted explosive (in “solid” state):

$$p_e = A_e \left(1 - \frac{\omega_e \eta_e}{R_{1e}}\right) e^{-\frac{R_{1e}}{\eta_e}} + B_e \left(1 - \frac{\omega_e \eta_e}{R_{2e}}\right) e^{-\frac{R_{2e}}{\eta_e}} + \omega_e \eta_e \rho_e E_e$$

where:

$$\eta_e = \rho_e / \rho_0$$

the relative density of the unreacted explosive.

$$E_e$$

the specified internal energy per unit mass of the unreacted explosive

$$\rho_0$$

the initial density of the explosive

$$A_e, B_e, \omega_e, R_{1e}, R_{2e}$$

the input constants of the unreacted explosive

Similarly, the pressure in the reaction products (in “gas” state) is defined by another JWL form as follows:

$$p_p^1 = A_p \left(1 - \frac{\omega_p \eta_p}{R_{1p}}\right) e^{-\frac{R_{1p}}{\eta_p}} + B_p \left(1 - \frac{\omega_p \eta_p}{R_{2p}}\right) e^{-\frac{R_{2p}}{\eta_p}} + \omega_p \eta_p \rho_p E_p$$

where:

$$\eta_p = \rho_p / \rho_0 = \text{the relative density of the unreacted explosive}$$

$$E_p = \text{the specified internal energy per unit mass of the unreacted explosive}$$

$$A_p, B_p, \omega_p, R_{1p}, R_{2p} = \text{the input constants of the reaction product. The chemical reaction rate for conversion of un-reacted explosive to reaction products is described by the following reaction rate equation:}$$

$$\frac{\partial F}{\partial t} = I(1 - F)^x (\eta_e - 1 - a)^r + G(1 - F)^x F^y (P)^z$$

here  $F$  denotes the burn fraction that is defined as the fraction of the explosive that has already reacted. For more details concerning the implementation of this equation of state, please refer to the Theory Manual

10. You can access the results of the un-reacted explosive and reaction products for IG elements. These EOSIG specific output variables are

| Keyword | Description   |
|---------|---|
| SIE-E   | Specific internal energy per unit mass of un-reacted explosive part |
| SIE-P   | Specific internal energy per unit mass of reaction products part    |
| FMAT    | Volume fraction   |
| RHO-E   | Density of un-reacted explosive part                                |
| RHO-P   | Density of reaction products part                                   |
| MASS-E  | Mass of un-reacted explosive part                                   |
| MASS-P  | Mass of reaction products part                                      |

The output variables for the burn fraction are

| Keyword | Type of Elements          | Description   |
|---------|---------------------------|---|
| FBURN   | Solid Lagrangian Elements | Burn fraction of EOSIG material                                       |
|         | Euler Elements            | Not applicable for EOSIG materials. Burn fraction for EOSJWL material |
| IGBURN  | Solid Lagrangian Elements | Not available   |
|         | Euler Elements            | Burn fraction of EOSIG MATERIAL                                       |

11. The ignition of IG material can be initiated by:

- Compression of the IG material in a small region, where the compression originates from outside that region. This is the most physical method to initiate ignition. Examples are a shock wave entering the region, a flow boundary that supplies mass to the region and a plate or other structural part that compresses the region. In all these cases the IG material should be initialized with zero pressures. This can be achieved by not specifying the specific energy on the TICVAL entry that prescribes the initial state of the IG material. The specific energy will be computed such that the initial pressure is zero.
- Compression of the IG material in a small region where the compression originates within that region. This can be done by specifying either a density that exceeds the compression limit or a specific energy that gives rise to a sufficiently large pressure.

Table 9-15 Coefficients for the IG Model of Several Explosions in the Database.

| Explosive  | PBX-9404<br>(a) | TATB     | PETN    | Case TNT | LANL<br>COMP B | Military<br>COMP B | PBX-9404<br>(b) | LX-17    |
|--|-----------------|----------|---------|----------|----------------|--------------------|-----------------|----------|
| Unreacted Equation of State and Constitutive Values: |                 |          |         |          |                |                    |                 |          |
| RHO<br>(g/cm <sup>3</sup> )                          | 1.842           | 1.90     | 1.842   | 1.61     | 1.712          | 1.630              | 1.842           | 1.903    |
| AE (Mbar)  | 69.69           | 108.2    | 37.42   | 17.98    | 778.1          | 1479.              | 9522.           | 778.1    |
| BE (Mbar)  | -1.727          | -2.406   | -1.313  | -0.931   | -0.05031       | -0.05261           | -0.5944         | -0.05031 |
| R1E  | 7.8             | 8.2      | 7.2     | 6.2      | 11.3           | 12.                | 14.1            | 11.3     |
| R2E  | 3.9             | 4.1      | 3.6     | 3.1      | 1.13           | 1.2                | 1.41            | 1.13     |
| OMGE   | 0.8578          | 1.251    | 1.173   | 0.8926   | 0.8938         | 0.9120             | 0.8867          | 0.8938   |
| Reacted Product Equation of State Values:            |                 |          |         |          |                |                    |                 |          |
| AP (Mbar)  | 8.524           | 6.5467   | 6.17    | 3.712    | 5.242          | 5.5748             | 8.524           | 6.5467   |
| BP (Mbar)  | 0.1802          | 0.071236 | 0.16926 | 0.032306 | 0.07678        | 0.0783             | 0.1802          | 0.071236 |
| R1P  | 4.6             | 4.45     | 4.4     | 4.15     | 4.2            | 4.5                | 4.6             | 4.45     |
| R2P  | 1.3             | 1.2      | 1.2     | 0.95     | 1.1            | 1.2                | 1.3             | 1.2      |
| OMGP   | 0.38            | 0.35     | 0.25    | 0.30     | 0.34           | 0.34               | 0.38            | 0.35     |
| ECHEM<br>(Mbar-cm <sup>3</sup> /g)                   | 0.0554          | 0.0363   | 0.0548  | 0.0433   | 0.0496         | 0.04969            | 0.0554          | 0.03626  |
| Reaction Rate Parameters:                            |                 |          |         |          |                |                    |                 |          |
| I ( $\mu$ s <sup>-1</sup> )                          | 44.0            | 50.0     | 20.0    | 50.0     | 44.0           | 44.0               | 44.0            | 50.0     |
| G (Mbar-z $\mu$ s <sup>-1</sup> )                    | 200.0           | 125.0    | 400.0   | 40.0     | 414.0          | 514.0              | 850.0           | 500.0    |
| A  | 0.01            | 0.01     | 0.01    | 0.01     | 0.01           | 0.01               | 0.01            | 0.0      |
| Z  | 1.6             | 2.0      | 1.4     | 1.2      | 2.0            | 2.0                | 2.0             | 3.0      |
| X  | 2/9             | 2/9      | 2/9     | 2/9      | 2/9            | 2/9                | 2/9             | 2/9      |
| Y  | 2/3             | 2/3      | 2/3     | 2/3      | 2/3            | 2/3                | 2/3             | 2/3      |
| R  | 4.0             | 4.0      | 4.0     | 4.0      | 4.0            | 4.0                | 4.0             | 4.0      |

12. EID must unique among all EOSxx entries in one model.

**EOSJWL****JWL Explosive Equation of State - SOL700**

Defines the properties of a JWL equation of state commonly used to calculate the pressure  $p$  of the detonation products of high explosives

$$p = p_0 + A \left(1 - \frac{\omega\eta}{R_1}\right) e^{-\frac{R_1}{\eta}} + B \left(1 - \frac{\omega\eta}{R_2}\right) e^{-\frac{R_2}{\eta}} + \omega\eta\rho_0 e + \frac{\omega\rho\lambda Q}{\rho_0}$$

$$\frac{d\lambda}{dt} = a(1 - \lambda)^m p^n$$

$e$  = specific internal energy per unit mass

$\rho_0$  = reference density

$\rho$  = overall material density

$\eta$  =  $\rho / \rho_0$

$p_0$  = initial pressure

$\lambda$  = fraction that describes the afterburning. It ranges from 0 to 1

A, B, R<sub>1</sub>, R<sub>2</sub>, a, m, n and Q are constants.

The last term:  $\frac{\omega\rho\lambda Q}{\rho_0}$  models afterburning.

### Format and Example

|                              |     |        |         |     |     |       |    |     |   |
|------------------------------|-----|--------|---------|-----|-----|-------|----|-----|---|
| EOSJWL                       | MID | A      | B       | R1  | R2  | OMEGA | P0 | Q   | + |
| EOSJWL                       | 37  | 5.2E11 | 0.77E11 | 4.1 | 1.1 | 0.34  |    | 0.0 | + |
| +      a      m      n       |     |        |         |     |     |       |    |     |   |
| +      0.0      0.0      0.0 |     |        |         |     |     |       |    |     |   |

| Descriptor | Meaning  | Type  | Default  |
|------------|--|-------|----------|
| EID        | Unique equation of state number referenced from MATDEUL. (Integer > 0; Required) | I > 0 | Required |
| A          | Constant A. (Real, 0.0)  | R     | 0.0      |
| B          | Constant B. (Real, 0.0)  | R     | 0.0      |
| R1         | Constant R1. (Real, 0.0)   | R     | 0.0      |
| P2         | Constant R2. (Real, 0.0)   | R     | 0.0      |

| Descriptor | Meaning                                     | Type         | Default            |
|------------|---|--------------|--------------------|
| OMEGA      | Constant $\omega$ . (Real, 0.0)             | R            | 0.0                |
| P0         | Initial pressure. See Remark 3. (Real, 0.0) | R            | 0.0; see Remark 4. |
| Q          | Initial pressure                            | $R \geq 0.0$ | 0.0; see Remark 6. |
| a          | Initial pressure                            | $R \geq 0.0$ | 0.0; see Remark 6. |
| m          | Initial pressure                            | $R \geq 0.0$ | 0.0; see Remark 6. |
| n          | Initial pressure                            | $R \geq 0.0$ | 0.0; see Remark 6. |

### Remarks:

1. This equation of state can be used only with Eulerian elements.
2. A [DETSPH](#) entry must be used to specify the detonation model.
3. This equation of state is discussed in *Dytran Theory Manual*, Chapter 4: Models, [EOSJWL – JWL Equation of State](#).
4. In simulations with ideal gases, the JWL material needs to have an initial pressure to counter balance the pressure of the ideal gas. Similarly, in case of under calculations where an explosive is located at a certain depth, P0 should be preset to equal the initial hydrostatic pressure.
5. The [DETSPH](#) definition of one EOSJWL material also applies to all other EOSJWL materials. Therefore, the blast wave of one explosive can ignite other explosives. To prevent this linked ignition, [PARAM,JWLDET,NOLINK](#) can be used.
6. By default the constants  $a, m, n$  and  $Q$  are zero, and afterburning is not taken into account. To model afterburning all the variables  $Q, a, m$  and  $n$  have to be set to positive constants. The extent of the afterburning can be checked by requesting the Euler element variable AFTERBURN. This variable equals  $\lambda$ . If the afterburning in an Euler element is complete then AFTERBURN = 1.0.

**EOSMG****Mie-Gruneisen Equation of State - SOL 700 only**

Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure  $p$  in high strain rate processes. Used in SOL 700 only.

$$p = \frac{\rho_0 c^2 n}{(1 - s\eta)^2} \left(1 - \frac{\Gamma_0 \eta}{2}\right) + \Gamma_0 \rho_0 e$$

$$\eta = 1 - \frac{\rho_0}{\rho_1}$$

$$\rho_1 = \min(\rho, RM)$$

where

|            |   |
|------------|---|
| $e$        | = specific internal energy per unit mass. For material at zero pressure, $e$ has to be initialized as zero.   |
| $\rho_0$   | = reference density   |
| $\rho$     | = overall material density  |
| $\Gamma_0$ | = Gruneisen parameter at reference density.   |
| $s$        | = defined by $U_s = c_0 + sU_p$ , where $U_s$ and $U_p$ are respectively the linear shock velocity and particle velocity as obtained from shock data. |
| $c$        | = sound speed at reference density  |
| RM         | = Cut-off value for density.  |

Format:

| 1     | 2   | 3 | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|-------|-----|---|---|----|----|---|---|---|----|
| EOSMG | EID | c | S | R1 | RM |   |   |   |    |

Example:

|       |    |      |     |     |      |  |  |  |  |
|-------|----|------|-----|-----|------|--|--|--|--|
| EOSMG | 37 | 2000 | 1.5 | 2.0 | 3000 |  |  |  |  |
|-------|----|------|-----|-----|------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique equation of state number referenced from MATDEUL. (Integer > 0; Required) |
| c          | Sound speed at reference density. (Real; Required)                               |
| s          | Constant $s$ . (Real; Required)  |
| $\Gamma_0$ | Gruneisen gamma. (Real; Required)  |
| RM         | Cut off value for density. (Real; Required)                                      |

**Remarks:**

1. This equation of state can be used only with Eulerian elements.
2. This equation of state is discussed in Equations of State.
3. The cut off value RM is only used for limiting the pressure. To prevent division by zero RM should be less than  $s/s - 1 \rho_{ref}$ . RM can be set slightly below this value. In case the simulation gets instable because of too large pressures RM can be decreased.
4. EID must unique among all EOSxx entries in one model.

**EOSNA****Noble-Abel equation of state - SOL700**

Defines the properties of Noble-Abel equation of state where the pressure  $p$  is defined as:

$$p = (\gamma - 1) \frac{\rho}{1 - b\rho} e$$

where:

- $e$  = specific internal energy per unit mass
- $\rho$  = overall material density
- $\gamma$  = a constant
- $b$  = Gas co-volume

**Format and Example**

| 1     | 2   | 3      | 4     | 5      | 6  | 7  | 8 | 9 | 10 |
|-------|-----|--------|-------|--------|----|----|---|---|----|
| EOSNA | EID | GAMMA  | B     | R      | CV | CP |   |   |    |
| EOSNA | 35  | 1.2363 | 0.001 | 314.46 |    |    |   |   |    |

| Field | Contents                           | Type    | Default                   |
|-------|------------------------------------|---------|---------------------------|
| EID   | Unique equation of state number.   | I > 0   | Required                  |
| GAMMA | Constant $\gamma$ .                | R ≥ 0.0 | Required                  |
| B     | Gas co-volume .                    | R ≥ 0.0 | 0.0<br>Refer to Remark 3. |
| R     | Gas constant.                      | R > 0.0 | Refer to remark 4.        |
| CV    | Specific heat at constant volume.  | R > 0.0 | Refer to remark 4.        |
| CP    | Specific heat at constant pressure | R > 0.0 | Refer to remark 4.        |

## Remarks

1. Refer to *Dytran Theory Manual*: Chapter 4-Models, for EOSNA – Noble-Abel Equation of State.
2. EOSNA – Noble-Abel Equation of State is used with Eulerian elements only.
3. The parameter B models the interactions between gas particles.
4. The temperature of the gas is calculated when one of the gas constants, R , C<sub>v</sub> , and C<sub>p</sub> is mentioned.
5. The Euler variable name for temperature is TEMPTURE.
6. The relation between Gamma, R , C<sub>v</sub> , and C<sub>p</sub> is shown below:

$$\gamma = \frac{C_p}{C_v} \quad R = C_p - C_v$$

**EOSPOL****Polynomial Equation of State for Solids - SOL 700 only**

Defines the properties of a polynomial equation of state where the pressure  $p$  is defined in SOL 700 as follows:

In compression ( $\mu > 0$ ),

$$p = a_1\mu + a_2\mu^2 + a_3\mu^3 + (b_0 + b_1 + b_2\mu^2 + b_3\mu^3)\rho_0e$$

In tension ( $\mu < 0$ ),

$$p = a_1\mu + (b_0 + b_1\mu)\rho_0e$$

Where

$$\mu = \eta - 1$$

$$\eta = \rho/\rho_0$$

$p$  = overall material density

$\rho_0$  = reference density

$e$  = specific internal energy per unit mass

**Used in SOL 700 only.**

**Format:**

| 1      | 2   | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|------|----|----|----|----|----|----|----|
| EOSPOL | EID | A1   | A2 | A3 | B0 | B1 | B2 | B3 |    |
|        | HVL | VISC |    |    |    |    |    |    |    |

**Example:**

|        |     |       |  |  |  |  |  |  |  |
|--------|-----|-------|--|--|--|--|--|--|--|
| EOSPOL | 100 | 80.E6 |  |  |  |  |  |  |  |
|        | 1.1 |       |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique equation of state member. (Integer > 0; Required) |
| A1         | Coefficient a1 or Bulk Modulus. (Real; Default = 0.0)    |
| A2         | Coefficient a2. (Real; Default = 0.0)                    |
| A3         | Coefficient a3. (Real; Default = 0.0)                    |
| B0         | Coefficient b0. (Real; Default = 0.0)                    |
| B1         | Coefficient b1. (Real; Default = 0.0)                    |
| B2         | Coefficient b2. (Real; Default = 0.0)                    |

| Descriptor | Meaning  |
|------------|--|
| B3         | Coefficient b3. (Real; Default = 0.0)                  |
| HVL        | Hydrodynamic volume limit. (Real > 1.0; Default = 1.1) |
| VISC       | Viscosity coefficient.(Real; Default = 0.0)            |

**Remarks:**

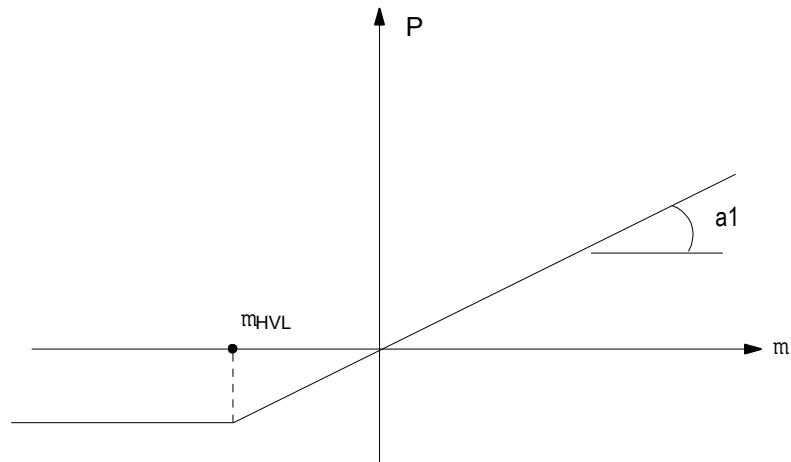
- When the relative volume ( $\rho_0/\rho$ ) exceeds HVL, the pressure is cut off to

$$P_{HVL} = f(\mu_{HVL})$$

with

$$\mu_{HVL} = \frac{1}{HVL} - 1$$

e.g., for  $p = a_1 \cdot \mu$ , the pressure behavior is as follows:



- When the PARAM,HVLFAIL is set to YES, the elements where the relative volume ( $\rho_0/\rho$ ) exceeds HVL fail completely. Their stress state is zero.

**EOSUDS****User-defined Equation of State - SOL 700 only**

Defines the frequency dependent properties for an isotropic poroelastic material. Use in SOL700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|------|---|---|---|---|----|
| EOSUDS | EID | GROUP | UNAME | VISC |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE water 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

| 1      | 2  | 3     | 4     | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|-----|---|---|---|---|----|
| EOSUDS | 12 | WATER | EXEOS | .01 |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique equation of state number. (Integer > 0; Required)                                   |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXEOS)                 |
| VISC       | Viscosity coefficient R > 0 No viscosity. See Remarks 4.and 5.                             |

**Remarks:**

1. The EID must be referenced by a MATDEUL or MAT1 entry.
2. The equation of state name is passed to method usrEOS of the SCAIMDSolver700 interface and can be used to identify the equation of state.
3. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient.
4. For the single mat solver viscous stresses will be stored in the output files by the use of TXX through TZx. Also, EFFSTS is available. For the multi-material solver viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZx-VIS. These viscous stresses only depend on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stresses. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.
5. UNAME can be:

| Subroutine Name | Function                                |
|-----------------|---|
| EXEOS           | Standard user defined Equation of State |

**EPOINT**

## Extra Point List

Defines extra points for use in dynamics problems.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| EPOINT | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 | ID7 | ID8 |    |

**Example:**

|        |   |    |   |   |    |   |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|
| EPOINT | 3 | 18 | 1 | 4 | 16 | 2 |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|

**Alternate Format and Example:**

|        |     |        |     |  |  |  |  |  |  |
|--------|-----|--------|-----|--|--|--|--|--|--|
| EPOINT | ID1 | "THRU" | ID2 |  |  |  |  |  |  |
| EPOINT | 17  | THRU   | 43  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| IDi        | Extra point identification number. (100,000,000 > Integer > 0; for "THRU" option, ID1 < ID2). |

**Remarks:**

1. All extra point identification numbers must be unique with respect to all other structural, scalar, and fluid points for direct methods of solution. For modal methods, they must be larger than the number of eigenvectors retained for analysis.
2. EPOINT is used to define coordinates used in transfer function definitions (see the TF and DMIG entries).
3. If the alternate format is used, extra points ID1 through ID2 are also defined to be extra points.
4. See the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of extra points.

**ERPPNL****Equivalent Radiated Power Definition**

Defines one or more panels by referencing sets of elements or properties.

**Format:**

| 1      | 2     | 3      | 4     | 5      | 6     | 7      | 8     | 9      | 10 |
|--------|-------|--------|-------|--------|-------|--------|-------|--------|----|
| ERPPNL | NAME1 | SETID1 | NAME2 | SETID2 | NAME3 | SETID3 | NAME4 | SETID4 |    |
|        | NAME5 | SETID5 |       |        |       |        |       |        |    |

**Example:**

|        |      |   |        |    |  |  |  |  |  |
|--------|------|---|--------|----|--|--|--|--|--|
| ERPPNL | ROOF | 1 | DOORLF | 16 |  |  |  |  |  |
|--------|------|---|--------|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NAMEi      | Panel label. (CHAR)  |
| SETIDi     | Identification number of a SET3 Bulk Data entry that lists the panel property entries or the panel elements. (Integer > 0) |

**Remarks:**

1. The SET3 entries can only refer to 2D and/or 3D structural element types, such as QUAD4, TRIA3, QUADR, CTRIAR, QUAD8, TRIA6, HEXA, PENTA and TETRA, or PSHELL, PCOMP, PCOMPG and PSOLID entry associated with those element types.
2. NAMEi are used in a Case Control SET definition defining *setp* to select the panels in the Case Control command ERP.
3. Duplicate Element IDs are removed if they occur on the referencing SET3 entries prior to the ERP calculation.
4. User does not need to specify the outer face(s) of 3D elements. The outer face(s) or exposed face(s) of 3D elements will be determined automatically. Interior 3D element(s) which has no exposed face(s) will not be included for ERP computation.

**EULFOR**

## Body Force Loading on Euler Elements - SOL 700 only

Defines a body force loading (acceleration) on Euler elements per unit mass. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3   | 4     | 5   | 6  | 7  | 8  | 9 | 10 |
|--------|-----|-----|-------|-----|----|----|----|---|----|
| EULFOR | BID | CID | VALUE | TID | N1 | N2 | N3 |   |    |

**Example:**

|        |     |   |  |    |    |    |    |  |  |
|--------|-----|---|--|----|----|----|----|--|--|
| EULFOR | 100 | 5 |  | 13 | 1. | 0. | 0. |  |  |
|--------|-----|---|--|----|----|----|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| BID        | Unique body force number. (Integer > 0; Required)   |
| CID        | ID of a coordinate system. (Integer $\geq 0$ ; Default = 0)   |
| VALUE      | Scale factor for the vector. See Remark 1. (Real $\geq 0.0$ )   |
| TID        | TABLED1 ID describing the scale factor for the load as function of time. See Remark 1. (Integer)  |
| N1, N2, N3 | Components of a vector giving the load (acceleration) direction defined in coordinate system CID. At least one must be nonzero. See Remark 2. (Real; Default = 0.0) |

**Remarks:**

1. Either VALUE or TID must be nonzero.
2. By default the components are zero, but at least one of them should be nonzero.

**EULFOR1**

Body force for Eulerian Regions - SOL700

Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.

Format:

|         |     |      |  |  |  |  |  |  |
|---------|-----|------|--|--|--|--|--|--|
| EULFOR1 | SID | ESID |  |  |  |  |  |  |
|---------|-----|------|--|--|--|--|--|--|

Example:

|         |     |   |  |  |  |  |  |  |
|---------|-----|---|--|--|--|--|--|--|
| EULFOR1 | 300 | 1 |  |  |  |  |  |  |
|---------|-----|---|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique EULFOR1 number referenced from a PEULER1 entry .(Integer >0, required) |
| ESID       | Group of geometric region EULFREG ID. (Integer > 0; Required)                 |

Remarks:

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.
4. To increase the accuracy of the region definition parameter MICRO can be used.

**EULFREG**

## Body force for Eulerian Regions - SOL700

Defines the acceleration field for sets of Eulerian regions. The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-depended acceleration can be defined.

**Format:**

|         |       |      |       |      |      |      |      |      |  |
|---------|-------|------|-------|------|------|------|------|------|--|
| EULFREG | ERID  | ESID | TYPE1 | VID1 | MID1 | ACCX | ACCY | ACCZ |  |
|         | LEVEL |      |       |      |      |      |      |      |  |

**Example:**

|         |     |   |     |     |     |     |     |     |  |
|---------|-----|---|-----|-----|-----|-----|-----|-----|--|
| EULFREG | 300 | 1 | BOX | 400 | 100 | 100 | 200 | 300 |  |
|         | 0.0 |   |     |     |     |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| ERID       | Unique ERID number. (Integer > 0; Required)   |
| ESID       | ID of group of Euler regions referenced from the EULFOR1 entry. (Integer > 0; Required)   |
| TYPEi      | The type of Eulerian region. (Character; Required)  |
|            | SURF      Region inside or outside a multifaceted surface.  |
|            | SPHERE    Region inside a sphere.   |
|            | CYLINDER Region inside a cylinder.  |
|            | BOX        Region inside a box.   |
|            | ELEM      Region defined by element list.   |
| VIDi       | Number of a geometric entity, a SET1 number, or number of a SURFINI entry. (Integer > 0; Required)  |
| MIDI       | Number of a MATDEUL entry to which the acceleration field will be applied. (Integer > 0; Required)  |
| ACCi       | Unique table number that defines the variation of acceleration in time. ACCX, ACCY and ACCZ respectively denote the acceleration in the x-, y- and z-direction. (Integer > 0; Required) |
| LEVELi     | Level indicator for this material and initial values. (Real; default=0.0)   |

**Remarks:**

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.

4. All level indicators LEVELi must have different values. The level indicator can be negative.
5. To increase the accuracy of the region definition parameter MICRO can be used.

**EXCLUDE**

Ignore Bulk Data entries from primary to secondary Module

Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6 | 7 | 8 | 9 | 10 |
|---------|--------|--------|--------|--------|---|---|---|---|----|
| EXCLUDE | ENTRY1 | ENTRY2 | ENTRY3 | -etc., |   |   |   |   |    |

**Example:**

|         |     |       |  |  |  |  |  |  |  |
|---------|-----|-------|--|--|--|--|--|--|--|
| EXCLUDE | SPC | FORCE |  |  |  |  |  |  |  |
|---------|-----|-------|--|--|--|--|--|--|--|

| Descriptor | Meaning                                  |
|------------|--|
| ENTRYi     | Name of a Bulk Data entry to be ignored. |

**Remarks:**

1. A secondary Module may have its own BEGIN MODULE section but it is not required as long as the user is willing to accept all modeling data from the reference Module because, by default, all entries in the reference Module will be copied to the secondary Module. However, if the user wishes to exclude and/or replace some of the entries from the reference Module in the copy then he simply specifies at least one instance of the entry to be ignored. For example, if the user wants to ignore all FORCE entries from the reference Module then specify a single FORCE entry in the secondary Module's Bulk Data section. Alternatively, the user may specify a list of Bulk Data entries to ignore on the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
2. EXCLUDE must be specified in the secondary (or copied) Module's Bulk Data section.
3. GRID entries cannot be ignored or replaced and EXCLUDE,GRID will cause a fatal error.
4. See [MDBULK](#) for the definition of secondary (or copied) Modules.

**EXTRN****Partitioned External Superelement Connection**

Defines a boundary connection for an external superelement.

**Format:**

| 1     | 2      | 3  | 4    | 5      | 6    | 7  | 8      | 9  | 10 |
|-------|--------|----|------|--------|------|----|--------|----|----|
| EXTRN | GID1   | C1 | GID2 | C2     | GID3 | C3 | GID4   | C4 |    |
|       | -etc.- |    | GID6 | "THRU" | GID7 | C6 | -etc.- |    |    |

**Example:**

|       |      |     |      |        |      |     |  |  |  |
|-------|------|-----|------|--------|------|-----|--|--|--|
| EXTRN | 1001 | 123 | 1120 | 123456 | 1201 | 123 |  |  |  |
|-------|------|-----|------|--------|------|-----|--|--|--|

| Descriptor       | Meaning  |
|------------------|--|
| GID <sub>i</sub> | Grid identification number to which the exterior superelement matrices will be connected.                              |
| C <sub>i</sub>   | Component numbers. (Integer 0, blank, or 1 for scalar points; Integers 1 through 6 with no embedded blanks for grids.) |

**Remarks:**

1. EXTRN can only be specified in partitioned Bulk Data Sections and is ignored in the main Bulk Data Section.
2. Connection grids must be specified in the partitioned Bulk Data Section following BEGIN SUPER = SEID.
3. "THRU" may be specified only in fields 3, 5, or 7.
4. Pairs of blank fields may be entered to allow easier modification of the EXTRN entry.
5. The order of the GID<sub>i</sub> and C<sub>i</sub> pairs must be in the internal order of the grids set in the creation run. But since Nastran no longer re-sequences grids then the pairs are specified in ascending GRID ID order. However, if PARAM,OLDSEQ or the SEQGP entry is specified then the proper order may not be ascending. To determine the internal grid order, add PARAM,USETPRT,0 and PARAM,USETSTR1,A.

## Entries F - L

### FAILJC

Johnson-Cook Failure Model - SOL 700 only

Defines the properties of the Johnson-Cook failure model. Used in SOL 700 only.

Format:

| 1      | 2     | 3  | 4   | 5  | 6  | 7  | 8                          | 9     | 10 |
|--------|-------|----|-----|----|----|----|----------------------------|-------|----|
| FAILJC | FID   | D1 | D2  | D3 | D4 | D5 | $\dot{\varepsilon}_{pl}^0$ | TROOM |    |
|        | TMELT | CP | MTH |    |    |    |                            |       |    |

Example:

|        |      |     |      |       |       |      |     |       |  |
|--------|------|-----|------|-------|-------|------|-----|-------|--|
| FAILJC | 1    | .05 | 3.44 | -2.12 | 0.002 | 0.16 | 1.0 | 297.0 |  |
|        | 1495 | 450 | CONT |       |       |      |     |       |  |

| Descriptor                 | Meaning   |
|----------------------------|---|
| FID                        | Unique failure model number. Referenced from MATDEUL. (Integer > 0; Required) |
| D1...D5                    | Parameters. See Remark 4. (Real; Default = 0.0)                               |
| $\dot{\varepsilon}_{pl}^0$ | Reference plastic strain rate. (Real; Default = 1.0)                          |
| TROOM                      | Room temperature. (Real; 0.0)   |
| TMELT                      | Melt temperature. (Real; 1.E+20)  |
| CP                         | Heat capacity. (Real; 1.E+20)   |
| MTH                        | Specifies how failure is applied. (Character; CONT)                           |
|                            | CONT      Continuous failure  |
|                            | DISC      Discrete failure  |
|                            | NOFAIL      Damage is not used for failure                                    |

Remarks:

1. This failure model is only available for Eulerian materials using the multi-material solver with strength.
2. The use of coupling surfaces is not supported.
3. The variable D can be visualized by adding DAMAGE to the Output request for Euler elements.
4. Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{time} \frac{\Delta \varepsilon_p}{\varepsilon^{frac}}$$

$$\varepsilon^{frac} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left( 1 + D_4 \ln \frac{\dot{\varepsilon}_{pl}}{\dot{\varepsilon}_{pl}^0} \right) (1 + D_5 T^*)$$

$$\sigma^* = \frac{\sigma_m}{\bar{\sigma}}$$

$$T^* = \frac{T - T_{room}}{T_{melt} - T_{room}}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature,  $\sigma$  is the mean stress,  $\sigma^*$  is the effective stress and  $\dot{\varepsilon}_{pl}$  is the fracture strain. The fracture strain depends on a non-dimensional plastic strain rate  $\dot{\varepsilon}_{pl}/\dot{\varepsilon}_{pl}^0$ . If D exceeds one it is set equal to one. The damage variable D is transported along with the Eulerian material.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

**FAILMPS**

## Maximum Plastic Strain Failure Model - SOL 700 only

Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value. Used in SOL 700 only.

Format:

|         |     |     |   |   |   |   |   |   |    |
|---------|-----|-----|---|---|---|---|---|---|----|
| 1       | 2   | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
| FAILMPS | FID | MPS |   |   |   |   |   |   |    |

Example:

|         |   |     |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|
| FAILMPS | 1 | .15 |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| FID        | Unique failure model number referenced from MATDEUL. (Integer > 0; Required) |
| MPS        | Maximum plastic strain that causes failure. (Real; Required)                 |

**FAILUDS****User Defined Failure For Nonlinear Explicit Analysis - SOL 700 only**

User defined simple failure of Eulerian materials. Use in SOL700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|---|---|---|---|---|----|
| FAILUDS | FID | GROUP | UNAME |   |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE usrfail 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

| 1       | 2   | 3       | 4      | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|---------|--------|---|---|---|---|---|----|
| FAILUDS | 200 | usrfail | EXFAIL |   |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| FID        | Unique output number. (Integer > 0; Required)  |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXFAIL)                |

**Remarks:**

1. Only can be used for SOL 700.
2. FID has to be referenced by a MATDEUL, MAT1 or MATORT entry.
3. UNAME can be:

| Subroutine Name | Function                           |
|-----------------|------------------------------------|
| EXFAIL          | Standard user defined failure      |
| EXFAIL1         | Alternative 1 user defined failure |
| EXFAIL2         | Alternative 2 user defined failure |

4. For option UNAME=EXFAIL2, for each material and for each Euler element a variable will be created that monitors the degree of failure of the material. This variable is denoted by DAMAGE and is between 0 and 1. The EXFAIL2 routine allows updating this damage variable due to the plastic strain increment of the current cycle.
5. For option UNAME=EXFAIL2 there are three ways in which this damage variable can model failure. These are:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1.0, the yield stress equals zero and the element fails.
  - Discrete failure: the element fails when D equals one.
  - No failure: positive damage values will not lead to failure. This is useful if the failure modeling is done by an YLDUDS routine. Then the yield stress can be reduced depending on the magnitude of the damage variable.
6. UNAME=EXFAIL2 is only supported by the multi-material Euler solver with strength.
  7. UNAME=EXFAIL1 is only supported with MATORT.
  8. For Lagrangian materials, UNAME=EXFAIL or EXFAIL1 are only available.

**FBADLAY****Dynamic Load Time Delay for FRF Based Assembly (FBA)**

Defines the time delay term  $\tau$  in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

| 1       | 2   | 3                   | 4     | 5 | 6     | 7 | 8 | 9 | 10 |
|---------|-----|---------------------|-------|---|-------|---|---|---|----|
| FBADLAY | SID | COMPID/<br>COMPNAME | PNTID | C | DELAY |   |   |   |    |

Examples:

|         |    |      |    |   |     |  |  |  |  |
|---------|----|------|----|---|-----|--|--|--|--|
| FBADLAY | 15 | BODY | 10 | 3 | 0.5 |  |  |  |  |
|---------|----|------|----|---|-----|--|--|--|--|

|         |    |    |   |   |     |  |  |  |  |
|---------|----|----|---|---|-----|--|--|--|--|
| FBADLAY | 25 | 30 | 5 | 2 | 0.1 |  |  |  |  |
|---------|----|----|---|---|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number. See Remark 1. (Integer > 0)   |
| COMPID     | Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)         |
| COMPNAME   | Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed) |
| PNTID      | Grid or scalar point identification number. See Remark 3. (Integer > 0)  |
| C          | Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)                                |
| DELAY      | Time delay term $\tau$ . See Remark 4. (Real)  |

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the time delay term  $\tau$  in frequency response analysis.
5. All FBADLAY entries specified in an FBA process are automatically converted internally by the program to equivalent DELAY entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

**FBALOAD**

## Load Scale Factor Specification for FRF Based Assembly (FBA) Process

Defines the scale factor for frequency dependent loads in an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2   | 3                   | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|---------------------|-------|---|---|---|---|---|----|
| FBALOAD | SID | COMPID/<br>COMPNAME | PNTID | C | A |   |   |   |    |

**Examples:**

|         |    |      |    |   |     |  |  |  |  |
|---------|----|------|----|---|-----|--|--|--|--|
| FBALOAD | 10 | WING | 20 | 3 | 2.5 |  |  |  |  |
| FBALOAD | 20 | 30   | 25 | 1 | 1.5 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number. See Remark 1. (Integer > 0)   |
| COMPID     | Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)                   |
| COMPNAME   | Optional name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed). |
| PNTID      | Grid or scalar point identification number. See Remark 3. (Integer > 0)  |
| C          | Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)  |
| A          | Scale factor. See Remark 4. (Real)   |

**Remarks:**

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the scale factor A in frequency response analysis.
5. All FBALOAD entries specified in an FBA process are automatically converted internally by the program to equivalent DAREA entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

**FBAPHAS****Dynamic Load Phase Lead for FRF Based Assembly (FBA)**

Defines the phase lead term  $\theta$  in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

| 1       | 2   | 3                   | 4     | 5 | 6     | 7 | 8 | 9 | 10 |
|---------|-----|---------------------|-------|---|-------|---|---|---|----|
| FBAPHAS | SID | COMPID/<br>COMPNAME | PNTID | C | PHASE |   |   |   |    |

Examples:

|         |    |       |    |   |      |  |  |  |  |
|---------|----|-------|----|---|------|--|--|--|--|
| FBAPHAS | 25 | FRAME | 40 | 1 | 10.0 |  |  |  |  |
| FBAPHAS | 30 | 50    | 10 | 2 | 5.0  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number. See Remark 1. (Integer > 0)   |
| COMPID     | Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)         |
| COMPNAME   | Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed) |
| PNTIDi     | Grid or scalar point identification numbers. See Remark 3. (Integer > 0)   |
| C          | Component number. See Remark 3. (Integers 1 through 6 for grid points; blank or 0 for scalar points.)                            |
| PHASE      | Phase lead term $\theta$ . See Remark 4. (Real)  |

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the phase lead term  $\theta$  in frequency response analysis.
5. All FBAPHAS entries specified in an FBA process are automatically converted internally by the program to equivalent DPHASE entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

**FBODYLD**

Equilibrated Free-Body Applied Load Case Definition

Defines an equilibrated free-body applied load case.

**Format:**

| 1       | 2     | 3       | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-------|---------|---|---|---|---|---|---|----|
| FBODYLD | NAMEL | FBODYSB |   |   |   |   |   |   |    |
|         | LABEL |         |   |   |   |   |   |   |    |

**Example:**

|         |                        |        |  |  |  |  |  |  |  |
|---------|------------------------|--------|--|--|--|--|--|--|--|
| FBODYLD | WINGLD                 | WINGSB |  |  |  |  |  |  |  |
|         | LOAD ON THE RIGHT WING |        |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| NAMEL      | User defined name identifying the load case. (Character; Required)  |
| FBODYSB    | Name of a FBODYSB Bulk Data entry that defines the subsystem for this load. (Character; Required)                                   |
| LABEL      | A string comprising no more than 64 characters (fields 2 through 9) that identifies and labels the load case. (Character; optional) |

**Remarks:**

1. NAMEL must be unique.
2. The Label is optional.

**FBODYSB****Equilibrated Free-Body Subsystems Definition**

Defines an equilibrated free-body subsystem.

**Format:**

| 1       | 2     | 3       | 4       | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|-------|---------|---------|-------|---|---|---|---|----|
| FBODYSB | NAMES | GRIDSET | ELEMSET | XFLAG |   |   |   |   |    |
|         | LABEL |         |         |       |   |   |   |   |    |

**Example:**

|         |            |   |   |     |  |  |  |  |  |
|---------|------------|---|---|-----|--|--|--|--|--|
| FBODYSB | WING       | 1 | 1 | ADM |  |  |  |  |  |
|         | RIGHT WING |   |   |     |  |  |  |  |  |

| Descriptor | Meaning   |   |
|------------|---|---|
| NAMES      | User defined name identifying the submodel. (Character; Required)   |   |
| GRIDSET    | Identification number of a SET1 entry that has a list of Grid Point Force grids to include in defining the subsystem. (Integer > 0) |   |
| ELEMSET    | Identification number of a SET1 entry that has a list of elements to include in the system (Integer > 0 or blank)                   |   |
| XFLAG      | Exclusion flag. Exclude the indicated Grid Point Force types.   |   |
|            | Default   | Blank (no type excluded)                        |
|            | S   | SPC forces                                      |
|            | M   | MPC forces                                      |
|            | A, L, or P  | Applied loads                                   |
|            | D   | DMIG's (and any other type not described above) |
| Label      | An optional string of up to 64 characters (fields 2 through 9) that identifies the subsystem.                                       |   |

**Remarks:**

1. Only those Grid Point Forces which have both an included grid point and element (or Grid Point Force type) will be taken into account.
2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
3. Fictitious grids or elements do not produce error or warning messages.
4. The XFLAG data can be any combination of the letters S,M,A,L,P and D (e.g., MAD).
5. The continuation is optional.

**FEEDGE****Finite Element Edge Definition**

Defines a finite element edge and associates it with a curve.

**Format:**

| 1      | 2      | 3     | 4     | 5     | 6      | 7   | 8   | 9 | 10 |
|--------|--------|-------|-------|-------|--------|-----|-----|---|----|
| FEEDGE | EDGEID | GRID1 | GRID2 | CIDBC | GEOMIN | ID1 | ID2 |   |    |

**Example:**

|        |     |     |     |  |        |    |  |  |  |
|--------|-----|-----|-----|--|--------|----|--|--|--|
| FEEDGE | 101 | 123 | 547 |  | GMCURV | 12 |  |  |  |
|--------|-----|-----|-----|--|--------|----|--|--|--|

| Descriptor | Meaning  |                  |          |  |
|------------|--|------------------|----------|--|
| EDGEID     | Unique identification number.  | Integer > 0      | Required |  |
| GRIDi      | Identification number of end GRIDs defining this edge.   | Integer > 0      | Required |  |
| CIDBC      | Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 1. | Integer $\geq 0$ | Required |  |
| GEOMIN     | Type of entry referenced by IDi; “GMCURV” or “POINT”. See Remark 2.  | Character        | POINT    |  |
| IDi        | Identification number of a POINT or GMCURV entry. See Remarks 2., 3., and 4.   | Integer $\geq 0$ |          |  |

**Remarks:**

- If CIDBC is not blank then it overrides the CIDBC specified on the GMSURF or FEFACE entries for this particular edge. A fatal message will be issued when more than one CIDBC is associated with any entity.
- The Bulk Data entries referenced by ID1 and ID2 depends on the GEOMIN field:

| GEOMIN | ID1    | ID2            |
|--------|--------|----------------|
| POINT  | POINT  | POINT          |
| GMCURV | GMCURV | not applicable |

- When GEOMIN = “GMCURV”

- FEEDGE associates the finite element model and the geometric information.
- GRID1 and GRID2 are the end points of the edge, and the edge is on the CURVID curve. A locally parametric cubic curve is fit to the geometric curve such that the two have the same tangent at GRIDi (C1 continuous).

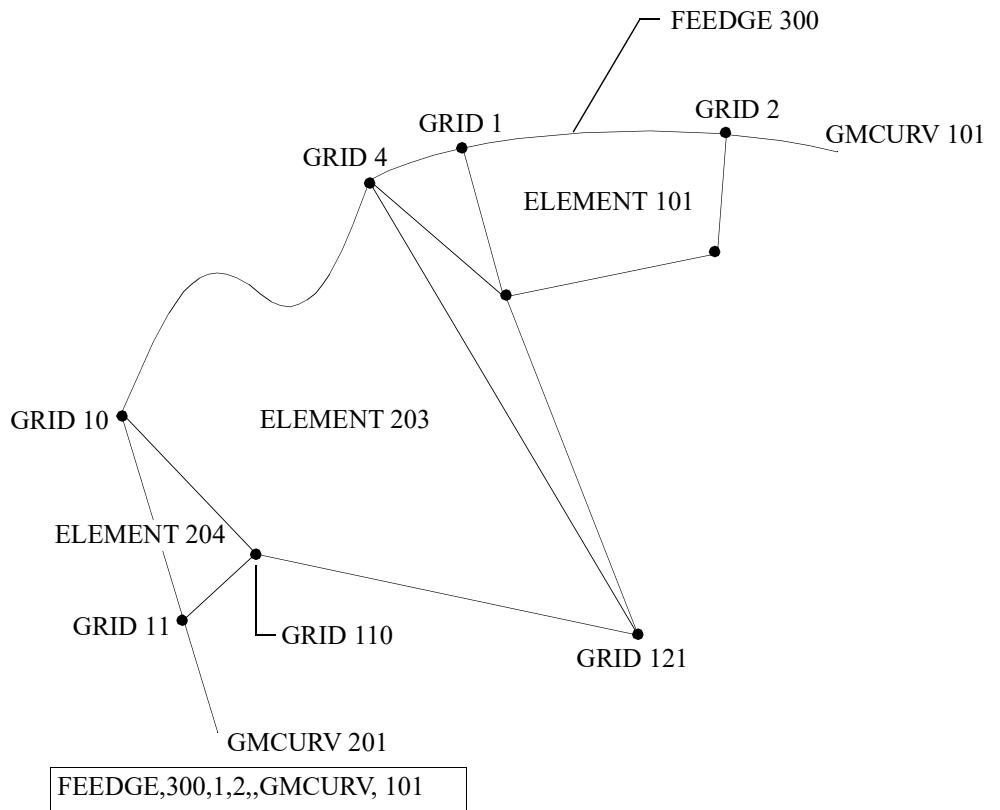


Figure 9-91 Specifying Geometry Using GEOMIN=GMCURV Method

4. When GEOMIN = "POINT"
  - The edge passes through the points defined on the POINT entries referenced by ID1 and ID2.
  - The shape of the edge is selected as follows:

| ID1        | ID2        | Shape of the FEEDGE |
|------------|------------|---------------------|
| Blank or 0 | Blank or 0 | Linear              |
| >0         | Blank or 0 | Quadratic           |
| >0         | >0         | Cubic               |
| Blank or 0 | >0         | Not allowed         |

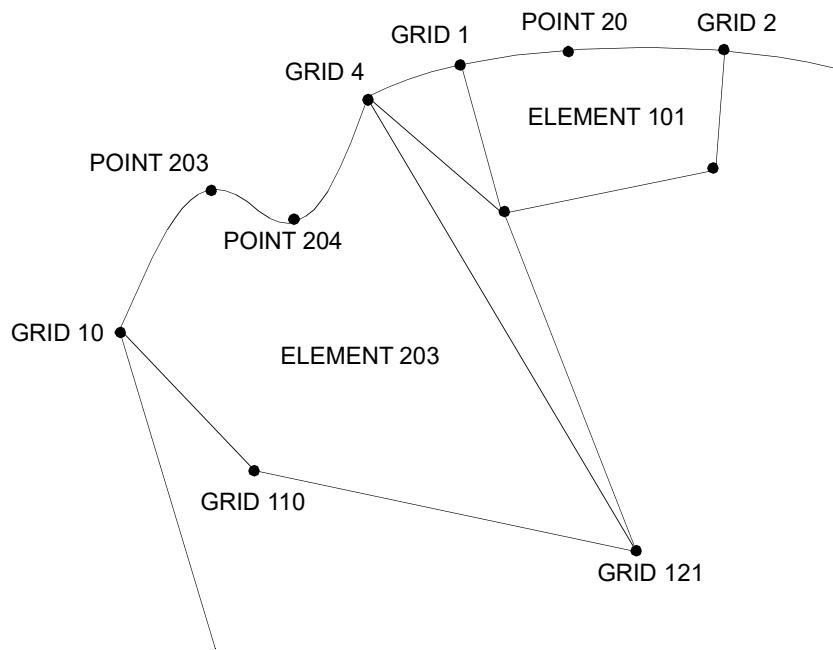


Figure 9-92 Specifying Geometry Using GEOMIN=POINT Method

5. A local coordinate system can be associated with an edge using the GMCORD entry.
6. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the [GMBC, 2072](#) entry description.

**FEFACE****Finite Element Face Definition**

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

Format:

| 1      | 2      | 3     | 4     | 5     | 6     | 7     | 8      | 9 | 10 |
|--------|--------|-------|-------|-------|-------|-------|--------|---|----|
| FEFACE | FACEID | GRID1 | GRID2 | GRID3 | GRID4 | CIDBC | SURFID |   |    |

Example:

|        |     |     |     |     |     |    |  |  |  |
|--------|-----|-----|-----|-----|-----|----|--|--|--|
| FEFACE | 101 | 123 | 547 | 243 | 295 | 12 |  |  |  |
|--------|-----|-----|-----|-----|-----|----|--|--|--|

| Descriptor | Meaning  |                  |           |
|------------|--|------------------|-----------|
| FACEID     | Unique identification number. See Remark 1.  | Integer > 0      | Required  |
| GRIDi      | Identification number of end GRIDs defining a triangular or quadrilateral face. See Remark 2.                                      | Integer > 0      | Required  |
| CIDBC      | Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 3. | Integer $\geq 0$ | Remark 3. |
| SURFID     | Alternate method used to specify the geometry of the edges of the face. See Remarks 4. and 5.                                      | Integer $\geq 0$ | 0         |

Remarks:

1. An FEFACE entry is required if any of the following situations exist:
  - The geometry of the surface defined by SURFID is to be used by a finite element;
  - CIDBC is specified for a face or surface; or
  - If loads or constraints or enforced boundary conditions are applied to a surface.
2. The shape (geometry) of the face is defined by the shape of the edges. The points defined by GRIDi must be specified in either a clockwise or counterclockwise order.
3. If CIDBC is not blank, then it overrides the CIDBC specified on the GMSURF entry for this particular face. A fatal message will be issued when more than one CIDBC is associated with any entity.
4. When SURFID is blank or 0, the edges will be considered linear unless there is an FEEDGE entry for the given edge.
5. When SURFID > 0,
  - FEFACE associates the finite element model and the geometric information specified on the GMSURF entry.

- GRIDi defines a finite element face (clockwise or counter clockwise in order) that is on the SURFID surface.
  - For the edges of this face, which are not defined by an FEEDGE entry, locally parametric cubic curves are fit to the geometric surface such that the two have the same tangent at GRIDi (C1 continuous).
6. Whenever a given edge of a face is common to two or more surfaces (i.e., lies on the intersecting curve), then the user must supply GMCURV and FEEDGE entries in order to resolve the conflict in the input geometry. A fatal message is issued if an edge is not uniquely defined.

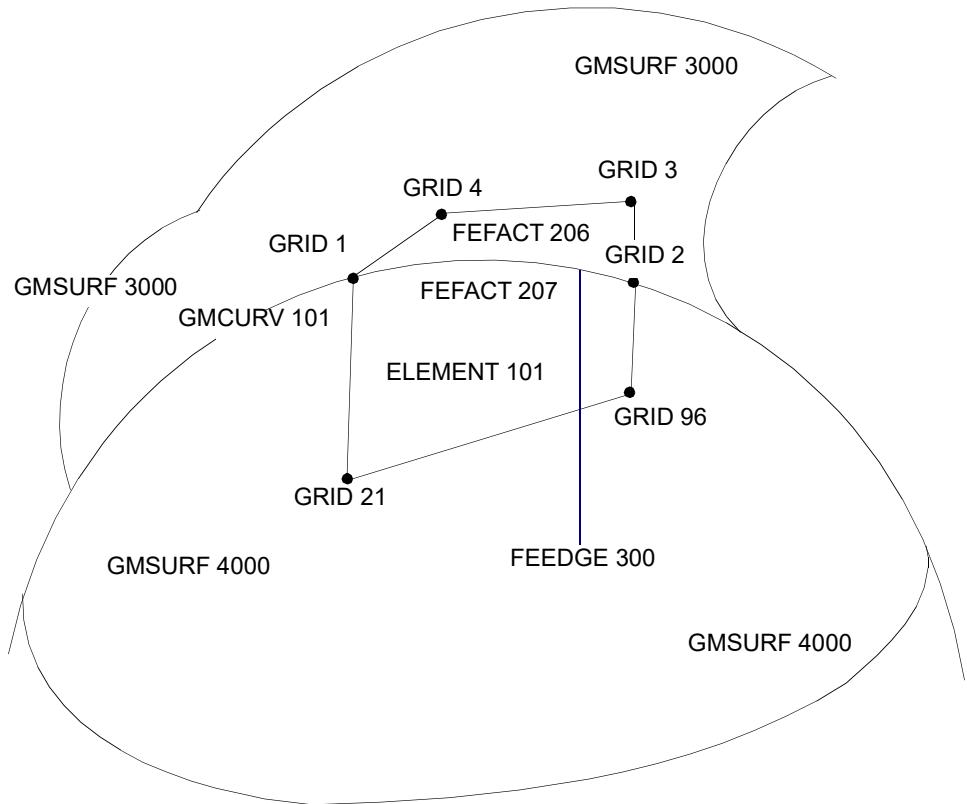


Figure 9-93 Face Edge Common to Two Surfaces

7. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the [GMBC, 2072](#) entry description.

**FFCONTR****Closed Volume Intended for Fluid Filled Containers**

Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling. Used in SOL 700 only.

Format:

| 1       | 2    | 3   | 4    | 5    | 6       | 7       | 8       | 9 | 10 |
|---------|------|-----|------|------|---------|---------|---------|---|----|
| FFCONTR | FFID | SID | FVOL | PATM | TEMPTAB | DENSTAB | TACTIVE |   |    |

Example:

|         |   |   |          |       |    |    |  |  |  |
|---------|---|---|----------|-------|----|----|--|--|--|
| FFCONTR | 1 | 2 | 1.50E-03 | 0.1E6 | 10 | 20 |  |  |  |
|---------|---|---|----------|-------|----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| FFID       | Unique FFCONTR identification number. (Integer > 0; Required)  |
| SID        | Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG entry defining the closed surface. (Integer > 0; Required)   |
| FVOL       | Fluid volume in the container. (Real $\geq 0.0$ ; Required)  |
| PATM       | Atmospheric pressure. Used for determination of the constant $C$ for $p \cdot V = C$ . (Real $> 0.0$ ; Required)   |
| TEMPTAB    | A reference to a TABLED1 ID that specifies how temperature of the container changes in time. (Integer $\geq 0$ ; Blank)  |
| DENSTAB    | A reference to a TABLED1 ID that specifies how density of the container changes with temperature. (Integer $\geq 0$ ; Blank)   |
| TACTIVE    | Time at which the pressure computation of the bottle is started. Until this time the pressure inside the bottle equals the ambient pressure. The volume of the bottle at TACTIVE will be used for the initial pressure computation of the gas in the bottle. |

Remarks:

1. If TEMPTAB is not set, the gas above the fluid is assumed to be an ideal, iso-thermal gas:  $p \cdot V = C$ , where  $C$  is a constant. If TEMPTAB is set, the temperature is applied to both the fluid as well as the gas. Then the gas satisfies  $p \cdot V/T = C$ , where  $T$  is the temperature of the fluid.
2. The fluid is assumed incompressible.
3. The pressure is based on the uniform pressure gasbag algorithm, where the pressure is uniform in the volume, but variable in time.
4. Output for the fluid-filled container is available through a SURFOUT definition. The available variables are: PRESSURE, VOLUME, TEMPTURE, VOLGAS, VOLFLUID, GAUGEPRES and RHOFLUID.
5. The normals of the surface referenced by SID are reversed automatically if required.

6. Modeling guidelines are described in the “Getting Started” Section.
7. If DENSTAB is set then volume of the fluid changes according to

$$V^{Fluid} = \frac{\rho(T_0)V_0^{Fluid}}{\rho(T)}$$

Here  $T_0$  and  $V_0^{Fluid}$  are initial values for temperature and fluid volume,  $\rho$  is the fluid density and  $T$  denotes the current temperature. If TEMPTAB is not set the DENSTAB entry will not be used.

8. At time=TACTIVE the gas is assumed to be in contact with the ambient pressure for the last time. This means that at Time = TACTIVE the pressure in the bottle equals the ambient pressure. After TACTIVE the bottle has been closed and there is no longer contact between ambient and gas inside the bottle. Any change in volume of the bottle or temperature or fluid will result in change of pressure of the gas inside the bottle.

**FLFACT****Aerodynamic Physical Data**

Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.

**Format:**

| 1      | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|--------|----|----|----|----|----|----|
| FLFACT | SID | F1 | F2     | F3 | F4 | F5 | F6 | F7 |    |
|        | F8  | F9 | -etc.- |    |    |    |    |    |    |

**Example:**

|        |    |    |    |     |  |  |  |  |  |
|--------|----|----|----|-----|--|--|--|--|--|
| FLFACT | 97 | .3 | .7 | 3.5 |  |  |  |  |  |
|--------|----|----|----|-----|--|--|--|--|--|

**Alternate Format and Example:**

|        |     |      |        |      |    |         |  |  |  |
|--------|-----|------|--------|------|----|---------|--|--|--|
| FLFACT | SID | F1   | "THRU" | FNF  | NF | FMID    |  |  |  |
| FLFACT | 201 | .200 | THRU   | .100 | 11 | .133333 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Unique Integer > 0)        |
| Fi         | Aerodynamic factor. (Real)                             |
| FNF        | Final aerodynamic factor. (Real)                       |
| NF         | Number of aerodynamic factors. (Integer > 0)           |
| FMID       | Intermediate aerodynamic factors. See Remark 4. (Real) |

**Remarks:**

- Only the factors selected by a FLUTTER entry will be used.
- Embedded blank fields are not allowed in the first format above.
- The factors must be specified in the order in which they are to be used within the looping of flutter analysis.
- $FMID$  must lie between  $F1$  and  $FNF$ ; otherwise,  $FMID$  will be set to  $(F1 + FNF)/2$ . Then

$$F_i = \frac{F1(FNF - FMID)(NF - i) + FNF(FMID - F1)(i-1)}{(FNF - FMID)(NF - i) + (FMID - F1)(i-1)}$$

where  $i = 1, 2, \dots, NF$

The use of  $FMID$  (middle factor selection) allows unequal spacing of the factors.

$FMID = \frac{2 \cdot F1 \cdot FNF}{F1 + FNF}$  gives equal values to increments of the reciprocal

of  $F_i$ .

5. If method = PK and this entry specifies velocities, then the velocities must be non-zero. Input of negative values produces eigenvector results at a velocity equal to the positive value of the input. Input of positive values provide eigenvalues results without eigenvectors.

**FLOW****Flow Boundary Condition**

Defines the properties of a material for the boundaries of an Eulerian mesh. Used in SOL 700 only.

Format:

| 1    | 2     | 3      | 4     | 5      | 6     | 7      | 8     | 9      | 10 |
|------|-------|--------|-------|--------|-------|--------|-------|--------|----|
| FLOW | LID   | BCID   | MESH  | DIR    |       |        |       |        |    |
|      | XMIN  | XMAX   | YMIN  | YMAX   | ZMIN  | ZMAX   |       |        |    |
|      | TYPE1 | VALUE1 | TYPE2 | VALUE2 | TYPE3 | VALUE3 | TYPE4 | VALUE4 |    |
|      | TYPE5 | VALUE5 |       |        |       |        |       |        |    |

Example:

|      |      |       |  |  |  |  |  |  |  |
|------|------|-------|--|--|--|--|--|--|--|
| FLOW | 120  | 122   |  |  |  |  |  |  |  |
|      |      |       |  |  |  |  |  |  |  |
|      | XVEL | 100.0 |  |  |  |  |  |  |  |
|      |      |       |  |  |  |  |  |  |  |

| Descriptor | Meaning   |   |
|------------|---|---|
| LID        | Number of a set of flow boundary conditions. (Integer > 0; Required)  |   |
| BCID       | Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 5. (Integer $\geq 0$ )  |   |
| MESH       | Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 6. (Integer $\geq 0$ )   |   |
| DIR        | Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 7. (Character)  |   |
| XMIN-ZMAX  | Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 8. (Real) |   |
| TYPEi      | The flow boundary property being defined. (Character)   |   |
|            | Material  | The MATDEUL ID number.                        |
|            | XVEL  | The material velocity in the x-direction.     |
|            | YVEL  | The material velocity in the y-direction.     |
|            | ZVEL  | The material velocity in the z-direction.     |
|            | PRESSURE  | The pressure of the material at the boundary. |
|            | DENSITY   | The density of the material at inflow.        |
|            | SIE   | The specific internal energy at inflow        |

| Descriptor | Meaning  |
|------------|--|
|            | FLOW                  The type of flow boundary required.  |
|            | HYDSTAT                A Hydrostatic pressure profile using a HYDSTAT entry.   |
| TYPEi      | The value for the property defined. (Real; Integer or Character; Required)<br><br>For TYPEi set to FLOW, the value is a character entry being either IN, OUT, BOTH or SYM defining that the flow boundary is defined as an inflow, outflow, possibly an in- or outflow or symmetry boundary. The default is BOTH.<br><br>TYPEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary.<br><br>For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used. |

**Remarks:**

1. LID must be referenced by a TLOAD1 entry.
2. Any material properties not specifically defined have the same value as the element with the flow boundary condition.
3. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
4. In the case of material flow into a multi-material Euler mesh, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
9. Prescribing both pressure and velocity may lead to the instabilities.
10. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

**FLOWC****Cyclic Flow Boundary Condition**

Defines the properties of a material for the boundaries of a Eulerian mesh.

Inflow values can be taken from another boundary condition. This allows cyclic or periodic boundary conditions. Likewise, the outflow of material goes into the other boundary condition.

FLOWC entries have to be defined in pairs. The FID on one entry has to be equal to FID2 of the other entry. For example

`FLOWC, 10, 30, 20`

`FLOWC, 20, 40, 10`

For FLOWC boundaries, mass flow summaries can be created as time history. Used in SOL700 only

Format:

| 1     | 2      | 3    | 4    | 5    | 6    | 7    | 8 | 9 | 10 |
|-------|--------|------|------|------|------|------|---|---|----|
| FLOWC | FID    | BCID | TYPE | MESH | DIR  | FID2 |   |   | +  |
| +     | XMIN   | XMAX | YMIN | YMAX | ZMIN | ZMAX |   |   | +  |
| +     | METHOD | TID  |      |      |      |      |   |   |    |
|       |        |      |      |      |      |      |   |   |    |

Example:

|       |   |     |   |  |  |  |  |  |  |
|-------|---|-----|---|--|--|--|--|--|--|
| FLOWC | 1 | 122 | 2 |  |  |  |  |  |  |
| +     |   |     |   |  |  |  |  |  |  |
| +     | 2 | 1   |   |  |  |  |  |  |  |
|       |   |     |   |  |  |  |  |  |  |

| Field | Contents   |
|-------|--|
| FID   | Unique number of a FLOWC entry. (Integer > 0; Required)  |
| BCID  | Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer > 0) |

| Field     | Contents  |
|-----------|---|
| TYPE      | <p>Flow boundary types. See Remarks 2.and 3. (Character; Required)</p> <p>IN: Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.</p> <p>OUT: Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</p> <p>BOTH: Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</p> |
| MESH      | Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer > 0)   |
| DIR       | Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character; no default)  |
| FID2      | Referenced FLOWC id from which inflow and outflow values will be taken from. (Integer > 0; Required)  |
| XMIN-ZMAX | Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real))  |
| METHOD    | <p>Method describing how material properties and pressure are mapped going from one boundary to the other. (Integer &gt; 0; Default=1)</p> <ol style="list-style-type: none"> <li>1. Map both velocity, material flow properties, and pressure loads one-to-one.</li> <li>2. Map velocity, material flow one-to-one. Pressure is given by element pressures.</li> <li>3. Inflow is taken as the average of outflow properties. This applies to both velocity and material properties. Pressure is given by element pressures.</li> </ol>  |
| TID       | TABLED1 ID that specifies a time dependent scale factor by which the amount of inflow is multiplied. This will not conserve total mass. (Integer > 0. Default = not used).  |

**Remarks:**

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.

5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. FLOWC can be used to specify flow boundaries for CHEXA's and also for Euler element created by the MESH,BOX option.
8. FLOWC is only supported by the multi-material Euler solver.
9. To enable using the outflow values of FID2 as the inflow values for FID, the boundary faces of FID and FID2 are mapped onto each other. For this mapping, only translations and rotations around coordinate axes are permitted.
10. Consider a cubic Euler mesh and that material flows from the left to the right. At the right side, the boundary condition is imposed

FLOWC,4,30,,5

and at the left side

FLOWC,5,40,,4

These definitions cause all material that flows out of the right side boundary into the left side boundary. Moreover, the Euler element pressures on the right side are put on the Euler elements of the left boundary condition.

In practice, it may be useful to skip the coupling between the two boundaries with regard to pressure. This can be done by setting METHOD equal to 2.

With these definitions, material from boundary 4 still flows into boundary 5 but pressure boundaries are transmitting.

11. TID is useful if several objects have identical outflow that is used as inflow by another object. Then only one object has to be modeled. To account for the other objects when defining inflow, the scale factor can be used. It can also be used to turn off in and outflow. When TID is set, either METHOD=2 or METHOD=3 are recommended and METHOD=1 should not be used.
12. It is allowed that the definition of the FLOWC entry overlaps with FLOW definitions. In that case, the FLOWC definition overrules the other ones

**FLOWDEF**

## Default Flow Boundary

Definition of default Eulerian flow boundary condition. Used in SOL 700 only.

Format:

| 1       | 2     | 3      | 4     | 5      | 6 | 7 | 8 | 9 | 10 |
|---------|-------|--------|-------|--------|---|---|---|---|----|
| FLOWDEF | FID   |        | TYPEM |        |   |   |   |   |    |
|         | TYPE1 | VALUE1 | TYPE2 | -etc.- |   |   |   |   |    |

Example:

|         |         |      |       |  |  |  |  |  |  |
|---------|---------|------|-------|--|--|--|--|--|--|
| FLOWDEF | 25      |      | HYDRO |  |  |  |  |  |  |
|         | DENSITY | 1000 |       |  |  |  |  |  |  |

| Descriptor | Meaning  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
|------------|--|----------|------------------------|------|---|------|---|------|---|----------|---|---------|--|-----|--|------|-------------------------------------|---------|---|
| FID        | Unique FLOWDEF number. (Integer > 0; Required)   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| TYPEM      | HYDRO, STRENGTH, MMHYDRO, or MMSTREN. (Character, HYDRO)   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| TYPEi      | The flow boundary property being defined. (Character) <table> <tr> <td>Material</td> <td>The MATDEUL ID number.</td> </tr> <tr> <td>XVEL</td> <td>The material velocity in the x-direction.</td> </tr> <tr> <td>YVEL</td> <td>The material velocity in the y-direction.</td> </tr> <tr> <td>ZVEL</td> <td>The material velocity in the z-direction.</td> </tr> <tr> <td>PRESSURE</td> <td>The pressure of the material at the boundary.</td> </tr> <tr> <td>DENSITY</td> <td>The density of the material at inflow.</td> </tr> <tr> <td>SIE</td> <td>The specific internal energy at inflow</td> </tr> <tr> <td>FLOW</td> <td>The type of flow boundary required.</td> </tr> <tr> <td>HYDSTAT</td> <td>A Hydrostatic pressure profile using a HYDSTAT entry.</td> </tr> </table> | Material | The MATDEUL ID number. | XVEL | The material velocity in the x-direction. | YVEL | The material velocity in the y-direction. | ZVEL | The material velocity in the z-direction. | PRESSURE | The pressure of the material at the boundary. | DENSITY | The density of the material at inflow. | SIE | The specific internal energy at inflow | FLOW | The type of flow boundary required. | HYDSTAT | A Hydrostatic pressure profile using a HYDSTAT entry. |
| Material   | The MATDEUL ID number.   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| XVEL       | The material velocity in the x-direction.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| YVEL       | The material velocity in the y-direction.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| ZVEL       | The material velocity in the z-direction.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| PRESSURE   | The pressure of the material at the boundary.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| DENSITY    | The density of the material at inflow.   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| SIE        | The specific internal energy at inflow   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| FLOW       | The type of flow boundary required.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| HYDSTAT    | A Hydrostatic pressure profile using a HYDSTAT entry.  |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |
| VALUEi     | The value for the property defined. (Real; Integer or Character; Required) <p>For TYPEi set to flow the value is a character entry being either IN, OUT, or BOTH, defining that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH.</p> <p>VALUEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary.</p> <p>For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used.</p>   |          |                        |      |   |      |   |      |   |          |   |         |  |     |  |      |                                     |         |   |

Remark:

1. If this entry is not specified, a default wall boundary condition is applied to all Eulerian free faces.
2. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

**FLOWT**

## Time Dependent Flow Boundary

Defines the material properties for the in- or outflow of material through the boundary of an Euler mesh. Inflow velocity and material properties can be chosen time dependent. Used in SOL 700 only.

Format:

| 1     | 2       | 3        | 4       | 5       | 6    | 7    | 8 | 9 | 10 |
|-------|---------|----------|---------|---------|------|------|---|---|----|
| FLOWT | FID     | BCID     | TYPE    | MESH    | DIR  |      |   |   |    |
|       | XMIN    | XMAX     | YMIN    | YMAX    | ZMIN | ZMAX |   |   |    |
|       | VELTYPE | VELOCITY | PRESTYP | PRES    |      |      |   |   |    |
|       | MID     | DENSTYP  | DENSITY | SIETYPE | SIE  |      |   |   |    |

Example:

|       |       |       |       |       |     |  |  |  |  |
|-------|-------|-------|-------|-------|-----|--|--|--|--|
| FLOWT | 2     | 122   | IN    |       |     |  |  |  |  |
|       |       |       |       |       |     |  |  |  |  |
|       | TABLE | 101   | TABLE | 102   |     |  |  |  |  |
|       | 91    | TABLE | 104   | TABLE | 107 |  |  |  |  |

| Descriptor | Meaning  |    |   |     |   |      |   |
|------------|--|----|---|-----|---|------|---|
| FID        | Unique number of a FLOWT entry. (Integer > 0; Required)  |    |   |     |   |      |   |
| BCID       | Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer $\geq 0$ )   |    |   |     |   |      |   |
| TYPE       | <table> <tr> <td>IN</td> <td>Inflow boundary. See Remarks 2 and 3. (Character; Required)<br/>Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.</td> </tr> <tr> <td>OUT</td> <td>Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</td> </tr> <tr> <td>BOTH</td> <td>Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</td> </tr> </table> | IN | Inflow boundary. See Remarks 2 and 3. (Character; Required)<br>Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie. | OUT | Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element. | BOTH | Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken. |
| IN         | Inflow boundary. See Remarks 2 and 3. (Character; Required)<br>Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.  |    |   |     |   |      |   |
| OUT        | Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.  |    |   |     |   |      |   |
| BOTH       | Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.  |    |   |     |   |      |   |
| MESH       | Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer $\geq 0$ )  |    |   |     |   |      |   |
| DIR        | Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character)   |    |   |     |   |      |   |

| Descriptor | Meaning  |
|------------|--|
| XMIN-ZMAX  | Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real)                                  |
| VELTYPE    | Type of velocity definition. (Character, Element)<br>ELEMENT Value of Euler Element<br>CONSTANT Value is constant in time<br>TABLE Value varies in time  |
| VELOCITY   | Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real) |
| PRESTYP    | Type of pressure definition (Character, Element)<br>ELEMENT Value of Euler element<br>CONSTANT Value is constant in time<br>TABLE Value varies in time   |
| PRES       | Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1 ID. (Integer or Real)  |
| MID        | MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and sie for the inflowing material. (Integer)  |
| DESTYP     | Type of density definition. (Character, Default is required when MID is given.)<br>ELEMENT Value of Euler element<br>CONSTANT Value is constant in time<br>TABLE Value varies in time  |
| DENSITY    | Value of density. If DENSTYP = TABLE it refers to a TABLED1 ID. (Character; Default is required when MID is given.)  |
| SIETYPE    | Type of density definition. (Character; Default is required when MID is given.)<br>ELEMENT Value of Euler element<br>CONSTANT Value is constant in time<br>TABLE Value varies in time.   |
| SIE        | Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1 ID. (Integer or Real; Default is required when MID is given.)   |

**Remarks:**

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.

3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. Any material properties not specifically defined have the same value as the element that with the boundary conditions.
8. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
9. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be instable. However, for supersonic inflow one can specify both pressure and velocities there are no outgoing waves at a supersonic inflow boundary.

**FLOWUDS**

User Defined Flow Boundary.

Defines a flow boundary on an Eulerian mesh specified by a user subroutine. Use in SOL700 only.

**Format:**

| 1       | 2    | 3     | 4     | 5    | 6    | 7    | 8 | 9 | 10 |
|---------|------|-------|-------|------|------|------|---|---|----|
| FLOWUDS | LID  | GROUP | UNAME | BCID | MESH | DIR  |   |   |    |
| +       | XMIN | XMAX  | YMIN  | YMAX | ZMIN | ZMAX |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myflow 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

*In Bulk Data:*

| 1       | 2  | 3      | 4      | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|-----|---|---|---|---|----|
| FLOWUDS | 12 | MYFLOW | EXFLOW | 300 |   |   |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| LID        | Unique output number. (Integer > 0; Required)   |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)  |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXFLOW)   |
| BCID       | Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 6. (Integer > 0)  |
| MESH       | Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 7. (Integer > 0)   |
| DIR        | Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 8. (Character)  |
| XMIN-ZMAX  | Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 9. (Real) |

**Remarks:**

1. Only can be used for SOL 700.
2. UNAME can be:

| Subroutine Name | Function  |
|-----------------|---|
| EXFLOW          | Standard user defined flow boundary on the Euler mesh |

3. For multi material models EXFLOW allows for the definition of any material to flow into the Eulerian mesh. The outflow can only be of materials present in the mesh.
4. TLOAD1 entries referencing FLOWUDS entries must have the TID field blank or zero.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.

**FLSYM****Axisymmetric Symmetry Control**

Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.

**Format:**

| 1     | 2 | 3  | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|---|----|----|---|---|---|---|---|----|
| FLSYM | M | S1 | S2 |   |   |   |   |   |    |

**Example:**

|       |    |   |   |  |  |  |  |  |  |
|-------|----|---|---|--|--|--|--|--|--|
| FLSYM | 12 | S | A |  |  |  |  |  |  |
|-------|----|---|---|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| M          | Number of symmetric sections of structural boundary around the circumference of the fluid being modeled by the set of structural elements. (Even Integers $\geq 2$ ) |
| S1, S2     | Description of boundary constraints used on the structure at the first and second planes of symmetry. (Character: "S" means symmetric, "A" means antisymmetric.)     |

**Remarks:**

1. This entry is allowed only if an AXIF entry is also present.
2. Only one FLSYM entry is allowed.
3. This entry is not required if there are no planes of symmetry.
4. First plane of symmetry is assumed to be at  $\phi = 0$ . Second plane of symmetry is assumed to be at  $\phi = 360^\circ/M$ .
5. Symmetric and antisymmetric constraints for the structure must, in addition, be provided by the user.
6. The solution is performed for those harmonic indices listed on the AXIF entry that are compatible with the symmetry conditions.
7. For example, if FLSYM is used to model a quarter section of structure at the boundary, M = 4. If the boundary constraints are "SS", the compatible cosine harmonics are 0, 2, 4, ..., etc. If "SA" is used, the compatible cosine harmonics are 1, 3, 5, ..., etc.

**FLUTTER****Aerodynamic Flutter Data**

Defines data needed to perform flutter analysis.

**Format:**

| 1       | 2   | 3      | 4    | 5    | 6     | 7     | 8               | 9   | 10 |
|---------|-----|--------|------|------|-------|-------|-----------------|-----|----|
| FLUTTER | SID | METHOD | DENS | MACH | RFREQ | IMETH | NVALUE/<br>OMAX | EPS |    |

**Example:**

|         |    |   |     |     |     |   |   |      |  |
|---------|----|---|-----|-----|-----|---|---|------|--|
| FLUTTER | 19 | K | 119 | 219 | 319 | S | 5 | 1.-4 |  |
|---------|----|---|-----|-----|-----|---|---|------|--|

| Descriptor     | Meaning  |
|----------------|--|
| SID            | Set identification number. (Integer > 0)   |
| METHOD         | Flutter analysis method. (Character: "K" for K method, "PK" for PK method, "PKNL" for PK method with no looping, "PKS" for PK sweep method, "PKNLS" for PK sweep method with no looping, "KE" for the K method restricted for efficiency.) See Remark 9. |
| DENS           | Identification number of an FLFACT entry specifying density ratios to be used in flutter analysis. (Integer > 0)   |
| MACH           | Identification number of an FLFACT entry specifying Mach numbers ( $m$ ) to be used in flutter analysis. (Integer > 0)   |
| RFREQ (or VEL) | Identification number of an FLFACT entry specifying reduced frequencies ( $k$ ) to be used in flutter analysis; for the "PKx" methods, the velocities FLFACT entry is specified in this field. (Integer > 0)   |
| IMETH          | Choice of interpolation method for aerodynamic matrix interpolation. See Remark 6. (Character: "L" = linear, "S" = surface; "TCUB" = termwise cubic; Default = "L".)   |
| NVALUE         | Number of eigenvalues beginning with the first eigenvalue for output and plots. (Integer > 0; Default is the number of modal degrees-of-freedom ( $u_h$ )).  |
| OMAX           | For the PKS and PKNLS methods, OMAX specifies the maximum frequency, in Hz., to be used in the flutter sweep. (Real > 0.0; Default = maximum normal mode eigenfrequency)   |
| EPS            | Convergence parameter for $k$ . Used in the PK and PKNL methods only. See Remark 4. (Real > 0.0; Default = $10^{-3}$ .) See Remark 9. for the meaning of EPS when the PKS or PKNLS methods are being used.   |

**Remarks:**

1. The FLUTTER entry must be selected with the Case Control command FMETHOD = SID.
2. The density is given by DENS · RHOREF, where RHOREF is the reference value specified on the AERO entry and DENS is the density ratio specified on the FLFACT entry.

3. The reduced frequency is given by  $k = (REFC \cdot \omega / 2 \cdot V)$ , where REFC is given on the AERO entry,  $\omega$  is the circular frequency, and  $V$  is the velocity. If  $k = 0.0$ , as specified on the FLFACT entry, then only the K method may be specified and the Inverse Power method of eigenvalue extraction (INV on the EIGC entry) must be used. Aeroelastic divergence analysis is more appropriately performed using one of the “PKx” methods.
4. For the PK and PKNL methods, an eigenvalue is accepted when:
 
$$|k - k_{estimate}| < EPS \quad \text{for } k_{estimate} < 1.0$$

$$|k - k_{estimate}| < EPS \cdot k_{estimate} \quad \text{for } k_{estimate} \geq 1.0$$
5. When one of the “PKx” methods is selected, physical displacements will only be generated for the velocities on the FLFACT that are specified as negative values of the requested velocity. Also, structural damping as specified on the GE field of MATi entries is ignored.
6. If IMETH = “L”, a linear interpolation is performed on reduced frequencies at the Mach numbers specified on the FLFACT entry using the MKAEROi entry Mach number that is closest to the FLFACT entry Mach number. For IMETH = “S”, a surface interpolation is performed across Mach numbers and reduced frequencies. For IMETH = “TCUB” a termwise cubic interpolation on reduced frequency is used. IMETH = “S” is only available for the “K” and “KE” flutter methods. IMETH = “TCUB” is only available for the “PKx” methods. For the “PKx” methods, IMETH = “S” or “L” or blank provides linear interpolation while “TCUB” provides a termwise cubic interpolation.
7. For the “K”, “KE”, “PK”, and “PKS” methods, all combinations of the FLFACT entry are analyzed. For the “PKNL” and “PKNLS” methods, only ordered pairs are analyzed; i.e.,  $(\rho_1, M_1, V_1), (\rho_2, M_2, V_2) \dots (\rho_n, M_n, V_n)$ . For the PKNL and PKNLS methods, equal number of densities, Mach numbers and velocities must be specified.
8. “K” and “KE” methods are not supported for design sensitivity and optimization.
9. The PKS and PKNLS methods determine flutter eigenvalues by performing a sweep of equally spaced reduced frequencies ranging from  $k_{est} = 0.0$  through  $k_{est} = \pi \cdot REFC \cdot OMAX / Velocity$ . The number of intervals is calculated using  $NINT = INT(1.0/EPS)$ .
10.
  - The PK method uses only real matrix terms for computing the flutter solution. This means any imaginary terms in any of the matrices, M, B, or K are ignored with the PK method and the imaginary part of the aerodynamic matrix is added as a real matrix to the viscous damping matrix B.
  - For the KE method, the B matrix is ignored while complex stiffness forms of structural damping are supported. Modal viscous damping (TABDMP1 input) will be included when the KDAMP parameter is set to -1.
  - The K method supports all forms of damping.
11. METHOD=K requires a CMETHOD in Case Control pointing to an EIGC Bulk Data entry. It is recommended that the number of roots requested on the EIGC entry be twice the number of normal modes used in the modal flutter analysis.

**FORCE****Static Force**

Defines a static concentrated force at a grid point by specifying a vector.

**Format:**

| 1     | 2   | 3 | 4   | 5 | 6  | 7  | 8  | 9 | 10 |
|-------|-----|---|-----|---|----|----|----|---|----|
| FORCE | SID | G | CID | F | N1 | N2 | N3 |   |    |

**Example:**

|       |   |   |   |     |     |     |     |  |  |
|-------|---|---|---|-----|-----|-----|-----|--|--|
| FORCE | 2 | 5 | 6 | 2.9 | 0.0 | 1.0 | 0.0 |  |  |
|-------|---|---|---|-----|-----|-----|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| G          | Grid point identification number. (Integer > 0)  |
| CID        | Coordinate system identification number. (Integer $\geq 0$ ; Default = 0)  |
| F          | Scale factor. (Real)   |
| Ni         | Components of a vector measured in coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$ . unless F is zero) |

**Remarks:**

1. The static force applied to grid point G is given by

$$\vec{f} = F \vec{N}$$

where  $\vec{N}$  is the vector defined in fields 6, 7 and 8. The magnitude of  $\vec{f}$  is equal to F times the magnitude of  $\vec{N}$ .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank (the default) references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body: the MID of a rigid material MATRIG or the EID of a RBE2. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2 and a rigid material are used with these TYPES. SOL 700 only.
6. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

**FORCE1****Follower Force, Alternate Form 1**

Defines a concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.

Format:

| 1      | 2   | 3 | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|--------|-----|---|---|----|----|---|---|---|----|
| FORCE1 | SID | G | F | G1 | G2 |   |   |   |    |

Example:

|        |   |    |       |    |    |  |  |  |  |
|--------|---|----|-------|----|----|--|--|--|--|
| FORCE1 | 6 | 13 | -2.93 | 16 | 13 |  |  |  |  |
|--------|---|----|-------|----|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)                                      |
| G          | Grid point identification number. (Integer > 0)                                    |
| F          | Magnitude of the force. (Real)   |
| G1, G2     | Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident.) |

Remarks:

1. The force applied to grid point G is given by

$$\vec{f} = F\vec{n}$$

where  $\vec{n}$  is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

**FORCE2**

## Follower Force, Alternate Form 2

Defines a concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

| 1      | 2   | 3 | 4 | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|-----|---|---|----|----|----|----|---|----|
| FORCE2 | SID | G | F | G1 | G2 | G3 | G4 |   |    |

Example:

|        |   |    |       |    |    |    |    |  |  |
|--------|---|----|-------|----|----|----|----|--|--|
| FORCE2 | 6 | 13 | -2.93 | 16 | 13 | 17 | 13 |  |  |
|--------|---|----|-------|----|----|----|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| G          | Grid point identification number. (Integer > 0)  |
| F          | Magnitude of the force. (Real)   |
| Gi         | Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident; G3 and G4 cannot be coincident.) |

Remarks:

1. The direction of the force is parallel to the cross product of vectors from G1 to G2 and G3 to G4.
2. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

**FORCEAX****Conical Shell Static Force**

Defines a static concentrated force on a conical shell ring.

**Format:**

| 1       | 2   | 3   | 4   | 5 | 6  | 7  | 8  | 9 | 10 |
|---------|-----|-----|-----|---|----|----|----|---|----|
| FORCEAX | SID | RID | HID | S | FR | FP | FZ |   |    |

**Example:**

|         |   |   |   |     |     |     |     |  |  |
|---------|---|---|---|-----|-----|-----|-----|--|--|
| FORCEAX | 1 | 2 | 3 | 2.0 | 0.1 | 0.2 | 0.3 |  |  |
|---------|---|---|---|-----|-----|-----|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| RID        | RINGAX entry identification number. (Integer > 0)   |
| HID        | Harmonic identification number or a sequence of harmonics. See Remark 5.<br>(Integer $\geq 0$ ) |
| S          | Scale factor for the force. (Real)  |
| FR, FP, FZ | Force components in $r, \phi, z$ directions. (Real)   |

**Remarks:**

1. FORCEAX is allowed only if an AXIC entry is also present.
2. Axisymmetric shell loads must be selected with the Case Control command LOAD = SID.
3. A separate entry is needed for the definition of the force associated with each harmonic.
4. See [Conical Shell Element \(RINGAX\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2" where n1 is the start of the sequence and n2 is the end of the sequence (e.g., for harmonics 0 through 10, the field would contain "S0T10").

**FORCUDS****User-defined Enforced Motion at Grid Points**

Defines enforced motion at grid points specified by a user subroutine. Used in SOL700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|----|---|---|---|---|----|
| FORCUDS | LID | GROUP | UNAME |    |   |   |   |   |    |
| +       | G1  | G2    | THRU  | G3 |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE extvel 'SCA.MDSolver.Obj.Uds.Dytran.Loads'
```

*In Bulk Data:*

|        |   |        |   |   |  |  |  |  |  |
|--------|---|--------|---|---|--|--|--|--|--|
| FORCUD | 1 | EXTVEL |   |   |  |  |  |  |  |
| +      | 1 | 2      | 3 | 5 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| LID        | Number of a set of loads.(Integer>0;required)  |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement.<br>(Character; required)  |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXTVEL)  |
| Gi         | Numbers of the grid points that are constrained. If the word THRU appears between two numbers, all the numbers in the range are included in the list. BY indicates the increment to be used within this range. (Integer or character; required.) |

**Remarks:**

1. LID must be referenced by a TLOAD1 entry.
2. FORCUDS can only be used to specify enforced velocities for grid points. The TYPE field on the TLOAD1 entry must be set to two. The TID on the TLOAD1 entry must be set to zero or blank (no time variation).
3. The GROUP name must be unique from all other GROUP names if more than one FORCUDS is used. Therefore each FORCUDS must have its own group name in a FMS section CONNECT SERVICE statement.
4. The constraint name is passed to the subroutine and can be used to identify the constraint.
5. A THRU specification, including the start and finish points in the range, must be on one line.
6. If the THRU specification is used, all the points in the sequence do not have to exist. Those that do not exist are ignored. The first point in the THRU specification must be a valid grid point. BY can be used to exclude grid points.

7. None of the fields in the list of grid points can be blank or zero, since this designation marks the end of the list.
8. Any number of continuation lines can be used to define the list of grid points.
9. UNAME can be:

| Subroutine Name | Function                               |
|-----------------|--|
| EXTVEL          | user defined velocities on grid points |

**FREEPT**

## Fluid Free Surface Point

Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|--------|-----|------|------|------|------|------|------|---|----|
| FREEPT | IDF | IDP1 | PHI1 | IDP2 | PHI2 | IDP3 | PHI3 |   |    |

**Example:**

|        |   |     |      |     |      |     |       |  |  |
|--------|---|-----|------|-----|------|-----|-------|--|--|
| FREEPT | 3 | 301 | 22.5 | 302 | 90.0 | 303 | 370.0 |  |  |
|--------|---|-----|------|-----|------|-----|-------|--|--|

| Descriptor | Meaning   |
|------------|---|
| IDF        | RINGFL entry identification number. (Integer > 0)                                       |
| IDPi       | Free surface point identification number. (Integer > 0)                                 |
| PHIi       | Azimuthal position on fluid point (RINGFL entry) in the fluid coordinate system. (Real) |

**Remarks:**

1. FREEPT is allowed only if an AXIF entry is also present.
2. All free surface point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The free surface points are used for the identification of output data only.
4. Three points may be defined on a single entry.
5. The referenced fluid point (IDF) must be included in a free surface list (FSLIST entry).
6. Output requests for velocity and acceleration can be made at these points.

**FREQ****Frequency List**

Defines a set of frequencies to be used in the solution of frequency response problems.

**Format:**

| 1    | 2   | 3  | 4   | 5      | 6  | 7  | 8  | 9  | 10 |
|------|-----|----|-----|--------|----|----|----|----|----|
| FREQ | SID | F1 | F2  | F3     | F4 | F5 | F6 | F7 |    |
|      | F8  | F9 | F10 | -etc.- |    |    |    |    |    |

**Example:**

|      |      |      |      |      |      |      |      |      |  |
|------|------|------|------|------|------|------|------|------|--|
| FREQ | 3    | 2.98 | 3.05 | 17.9 | 21.3 | 25.6 | 28.8 | 31.2 |  |
|      | 29.2 | 22.4 | 19.3 |      |      |      |      |      |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)                             |
| Fi         | Frequency value in units of cycles per unit time. (Real $\geq 0.0$ ) |

**Remarks:**

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FREQ1**

## Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

Format:

| 1     | 2   | 3  | 4  | 5   | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----|----|-----|---|---|---|---|----|
| FREQ1 | SID | F1 | DF | NDF |   |   |   |   |    |

Example:

|       |   |     |     |    |  |  |  |  |  |
|-------|---|-----|-----|----|--|--|--|--|--|
| FREQ1 | 6 | 2.9 | 0.5 | 13 |  |  |  |  |  |
|-------|---|-----|-----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)                      |
| F1         | First frequency in set. (Real $\geq 0.0$ )                    |
| DF         | Frequency increment. (Real $> 0.0$ )                          |
| NDF        | Number of frequency increments. (Integer $> 0$ ; Default = 1) |

Remarks:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 + DF \cdot (i-1)$$

where i = 1 to (NDF + 1).

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FREQ2****Frequency List, Alternate Form 2**

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

Format:

| 1     | 2   | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----|----|----|---|---|---|---|----|
| FREQ2 | SID | F1 | F2 | NF |   |   |   |   |    |

Example:

|       |   |     |     |   |  |  |  |  |  |
|-------|---|-----|-----|---|--|--|--|--|--|
| FREQ2 | 6 | 1.0 | 8.0 | 6 |  |  |  |  |  |
|-------|---|-----|-----|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)                    |
| F1         | First frequency. (Real > 0.0)                               |
| F2         | Last frequency. (Real > 0.0, F2 > F1)                       |
| NF         | Number of logarithmic intervals. (Integer > 0; Default = 1) |

Remarks:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 \cdot e^{(i-1)d}$$

where  $d = \frac{1}{NF} \ln \frac{F2}{F1}$  and  $i = 1, 2, \dots, (NF + 1)$

In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}| ,$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FREQ3****Frequency List, Alternate 3**

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

Format:

| 1     | 2   | 3  | 4  | 5    | 6   | 7       | 8 | 9 | 10 |
|-------|-----|----|----|------|-----|---------|---|---|----|
| FREQ3 | SID | F1 | F2 | TYPE | NEF | CLUSTER |   |   |    |

Example:

|       |   |      |        |        |    |     |  |  |  |
|-------|---|------|--------|--------|----|-----|--|--|--|
| FREQ3 | 6 | 20.0 | 2000.0 | LINEAR | 10 | 2.0 |  |  |  |
|-------|---|------|--------|--------|----|-----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)  |
| F1         | Lower bound of modal frequency range in cycles per unit time. (Real $\geq 0.0$ for TYPE = LINEAR and Real = 0.0 for TYPE = LOG)   |
| F2         | Upper bound of modal frequency range in cycles per unit time. (Real $> 0.0$ ; F2 $\geq$ F1; Default = F1)   |
| TYPE       | LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")   |
| NEF        | Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1; Default = 10) |
| CLUSTER    | Specifies clustering of the excitation frequency near the end points of the range. See Remark 6. (Real > 0.0; Default = 1.0)  |

Remarks:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.
3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.
4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

5. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

6. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER > 1.0 provides closer spacing of excitation frequency towards the ends of the frequency range, while values of less than 1.0 provide closer spacing towards the center of the frequency range. For example, if the frequency range is between 10 and 20, NEF = 11, TYPE = "LINEAR"; then, the excitation frequencies for various values of CLUSTER would be as shown in [Table 17](#).

$$\hat{f}_k = \frac{1}{2}(\hat{f}_1 + \hat{f}_2) + \frac{1}{2}(\hat{f}_2 - \hat{f}_1)|\xi|^{1/\text{CLUSTER}} \cdot \text{SIGN}(\xi)$$

where

$\xi = -1 + 2(k - 1)/(NEF - 1)$  is a parametric coordinate between -1 and 1

$k$  = varies from 1 to NEF ( $k = 1, 2, \dots, NEF$ )

$\hat{f}_1$  = is the lower limit of the frequency subrange

$\hat{f}_2$  = is the upper limit of the subrange

$\hat{f}_k$  = is the k-th excitation frequency

$\hat{f}$  = is the frequency, or the logarithm of the frequency, depending on the value specified for TYPE

Table 17 CLUSTER Usage Example

| Excitation Frequency Number | $\xi$ | CLUSTER                         |        |       |       |       |
|-----------------------------|-------|---------------------------------|--------|-------|-------|-------|
|                             |       | c=0.25                          | c-0.50 | c-1.0 | c-2.0 | c-4.0 |
|                             |       | Excitation Frequencies in Hertz |        |       |       |       |
| 1                           | -1.0  | 10.00                           | 10.0   | 10.0  | 10.00 | 10.00 |
| 2                           | -0.8  | 12.95                           | 11.8   | 11.0  | 10.53 | 10.27 |
| 3                           | -0.6  | 14.35                           | 13.2   | 12.0  | 11.13 | 10.60 |
| 4                           | -0.4  | 14.87                           | 14.2   | 13.0  | 11.84 | 11.02 |
| 5                           | -0.2  | 14.99                           | 14.8   | 14.0  | 12.76 | 11.66 |
| 6                           | 0.0   | 15.00                           | 15.0   | 15.0  | 15.00 | 15.00 |
| 7                           | 0.2   | 15.01                           | 15.2   | 16.0  | 17.24 | 18.34 |

Table 17 CLUSTER Usage Example

| Excitation Frequency Number | $\xi$ | CLUSTER                         |        |       |       |       |
|-----------------------------|-------|---------------------------------|--------|-------|-------|-------|
|                             |       | c=0.25                          | c=0.50 | c=1.0 | c=2.0 | c=4.0 |
|                             |       | Excitation Frequencies in Hertz |        |       |       |       |
| 8                           | 0.4   | 15.13                           | 15.8   | 17.0  | 18.16 | 18.98 |
| 9                           | 0.6   | 15.65                           | 16.8   | 18.0  | 18.87 | 19.40 |
| 10                          | 0.8   | 17.05                           | 18.2   | 19.0  | 19.47 | 19.73 |
| 11                          | 1.0   | 20.00                           | 20.0   | 20.0  | 20.00 | 20.00 |

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FREQ4**

## Frequency List, Alternate Form 4

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of “spread” around each natural frequency and the number of evenly spaced excitation frequencies within the spread.

Format:

| 1     | 2   | 3  | 4  | 5    | 6   | 7 | 8 | 9 | 10 |
|-------|-----|----|----|------|-----|---|---|---|----|
| FREQ4 | SID | F1 | F2 | FSPD | NFM |   |   |   |    |

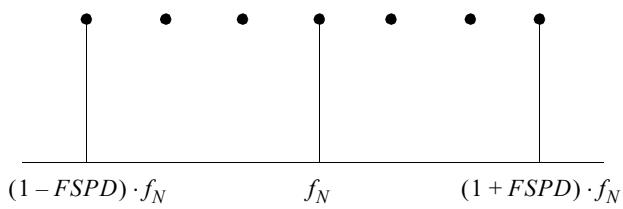
Example:

|       |   |      |        |      |    |  |  |  |  |
|-------|---|------|--------|------|----|--|--|--|--|
| FREQ4 | 6 | 20.0 | 2000.0 | 0.30 | 21 |  |  |  |  |
|-------|---|------|--------|------|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| F1         | Lower bound of frequency range in cycles per unit time. (Real $\geq 0.0$ ; Default = 0.0)  |
| F2         | Upper bound of frequency range in cycles per unit time. (Real $> 0.0$ ; F2 > F1; Default = 1.0E20)   |
| FSPD       | Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0; Default = 0.10) |
| NFM        | Number of evenly spaced frequencies per “spread” mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)                             |

Remarks:

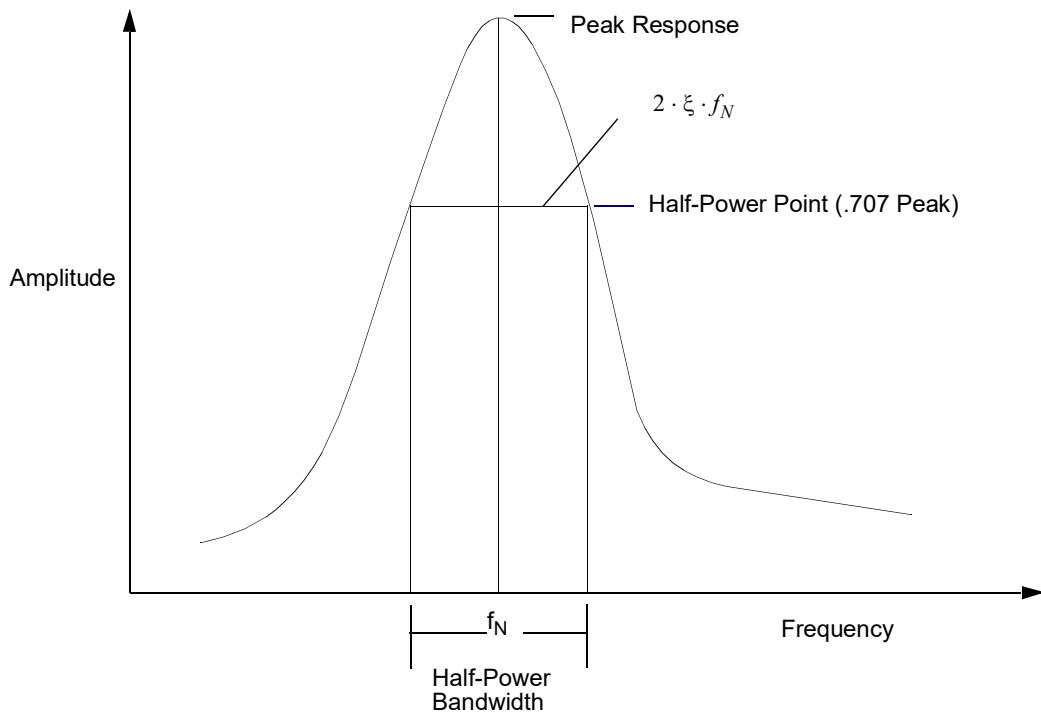
1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200 and is ignored in direct frequency-response solutions.
2. FREQ4 entries must be selected with the Case Control command FREQUENCY = SID.
3. There will be NFM excitation frequencies between  $(1 - FSPD) \cdot f_N$  and  $(1 + FSPD) \cdot f_N$ , for each natural frequency in the range F1 to F2.
4. In the example above there will be 21 equally spaced frequencies across a frequency band of  $0.7 \cdot f_N$  to  $1.3 \cdot f_N$  for each natural frequency that occurs between 20 and 2000. See [Figure 9-94](#) for the definition of frequency spread.



**Figure 9-94 Frequency Spread Definition**

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

- The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by  $2 \cdot \xi \cdot f_N$ , where  $\xi$  is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See [Figure 9-95](#) for the definition of half-power bandwidth.

**Figure 9-95 Half-Power Bandwidth Definition**

- Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.
- All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ . The values  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FREQ5****Frequency List, Alternate Form 5**

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

**Format:**

| 1     | 2   | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|--------|-----|-----|-----|-----|-----|----|
| FREQ5 | SID | F1  | F2     | FR1 | FR2 | FR3 | FR4 | FR5 |    |
|       | FR6 | FR7 | -etc.- |     |     |     |     |     |    |

**Example:**

|       |      |      |        |     |     |     |     |      |  |
|-------|------|------|--------|-----|-----|-----|-----|------|--|
| FREQ5 | 6    | 20.0 | 2000.0 | 1.0 | 0.6 | 0.8 | 0.9 | 0.95 |  |
|       | 1.05 | 1.1  | 1.2    |     |     |     |     |      |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| F1         | Lower bound of frequency range in cycles per unit time. (Real $\geq 0.0$ ; Default = 0.0)          |
| F2         | Upper bound of frequency range in cycles per unit time. (Real $> 0.0$ ; F2 > F1; Default = 1.0E20) |
| FRi        | Fractions of the natural frequencies in the range F1 to F2. (Real $> 0.0$ )                        |

**Remarks:**

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ5 entries must be selected with the Case Control command FREQUENCY = SID.
3. The frequencies defined by this entry are given by

$$f_i = FRi \cdot f_{N_i}$$

where  $f_{N_i}$  are the natural frequencies in the range F1 through F2.

4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less than F1 and greater than F2, those computed excitation frequencies are ignored.

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ . The values  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FRFCOMP****Frequency Response Function (FRF) Component Specification for FRF Based Assembly (FBA)**

Specifies the FRF components that are to be assembled as part of an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2      | 3        | 4      | 5      | 6        | 7        | 8 | 9 | 10 |
|---------|--------|----------|--------|--------|----------|----------|---|---|----|
| FRFCOMP | COMPID | COMPNAME | MEDIUM | UNITNO | LSCALFAC | FSCALFAC |   |   |    |

**Examples:**

|         |    |          |     |    |        |  |  |  |  |
|---------|----|----------|-----|----|--------|--|--|--|--|
| FRFCOMP | 10 | LEFTWING | OP2 | 25 |        |  |  |  |  |
| FRFCOMP | 20 | FRAME    | UF  | 30 | 1000.0 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| COMPID     | ID of the component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)   |
| COMPNAME   | Name of the COMPID FRF component. See Remark 1. (Up to 8 characters).   |
| MEDIUM     | Medium on which the FRF matrices and other related data are stored. Acceptable character values are DB (for database), OP2 (for OUTPUT2 file) or UF (for Universal file). See Remarks 2., 3. and 4. |
| UNITNO     | Fortran unit number for the OP2 and UF options. (Integer > 0). See Remarks 3. and 4.  |
| LSCALFAC   | Length scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10. and 11.   |
| FSCALFAC   | Force scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10., and 11.   |

**Remarks:**

1. COMPNAME may be referenced by the ASMOUT keyword in the FRF Case Control command in order to request output for a specific FRF component in the FBA process. Accordingly, COMPNAME may not have the values of CONNINFO, ALL, COMP or ASSEMBLY as these are all reserved words for use with the ASMOUT keyword in the FRF Case Control command.
2. If the DB option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the database for the specified FRF component:  

```
ASSIGN dbname = 'frfgen_job.MASTER'
DBLOCATE DATABLK = (FRFDB) LOGICAL = dbname
```

3. If the OP2 option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the OUTPUT2 file for the specified FRF component:  
ASSIGN INPUTT2 = 'frfgen\_job\_op2' UNIT = 25
4. If the UF option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the Universal file for the specified FRF component:  
ASSIGN UNVFILE = 'frfgen\_job\_unv' UNIT = 26
5. Component ID of 0 is assigned to the assembled FRF configuration resulting from the FBA process.
6. An FRF generation job using an FRF Case Control command with GEN/GENASM and COMPID and COMPNAME keywords specified in it will automatically generate and save an FRFCOMP Bulk Data entry on the assembly punch (.asm) file for that FRF component for subsequent use in an FBA process.
7. LSCALFAC and FSCALFAC are meaningful only for the UF option. They are ignored for the DB and OP2 options.
8. The FRF and other information on the Universal File (UF) is grouped by so-called Universal Dataset Numbers (UDNs). The heart of the information on the UF for a test FRF component is in UDN 58 which contains FRF data for that component. Also of interest, if present, is UDN 15 which contains grid point coordinate data in single precision or UDN 2411 which contains grid point coordinate data in double precision. Further, if UDN 15 or UDN 2411 is present, the data therein may reference definition or displacement coordinate systems. Such coordinate system data is resident in UDN 18.  
  
Details of the various UDNs and their formats can be obtained from the following websites:  
<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1>  
<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1/file-format-storehouse/file-formats>
9. The points of a test FRF component that are considered by the FBA process are those that are defined in UDN 58 either as response points or as excitation points. Any such point that appears both as a response point and as an excitation point is regarded as a potential connection point of the test FRF component.
10. Connections between test FRF components and other FRF components are determined by what is supplied on the UF as indicated below.
  - a. If UDN 15 or UDN 2411 is supplied on the UF, then connections between the test FRF component grid points and those of other FRF components will be based on *matching coordinates*.
  - b. If neither UDN 15 nor UDN 2411 is supplied, then normally, connections between the test FRF component grid points and those of other FRF components will be based on *matching grid point IDs*. However, the user can override this by specifying FRFCCONN entries, thereby causing connections between *user specified grid points*. In any case, when the connections are based on matching grid point IDs or based on FRFCCONN entries, it is the user's responsibility to ensure that the connection grid points have the same physical locations in space so that correct results are obtained from the FBA process. The program clearly cannot check for this condition since the coordinates of one or more of the points are not available.
11. For the UF option, LSCALFAC and FSCALFAC are used as follows:

- For Universal Dataset No. (UDN) 15 or 2411 (if present) on the Universal File:  
The X, Y, and Z coordinate values in the UDN are multiplied by LSCALFAC before they are used in the FBA process.
- For Universal Dataset No. (UDN) 58 on the Universal File:  
The FRF response quantities in the UDN are multiplied by appropriate factors as shown in the following table before they are used in the FBA process.

Table 9-16 Multiplication Factors for FRF Response Quantities in UDN 58 of the Universal File

| FRF Response Quantity  | Type of Excitation                 | Factor by Which FRF Response Quantity is Multiplied Before It is Used in the FBA Process |
|--|------------------------------------|--|
| Translational displacement, velocity or acceleration at a structural point | Unit force at an excitation point  | LSCALFAC/FSCALFAC  |
|  | Unit moment at an excitation point | 1.0/FSCALFAC   |
| Rotational displacement, velocity or acceleration at a structural point    | Unit force at an excitation point  | 1.0/FSCALFAC   |
|  | Unit moment at an excitation point | 1.0/(LSCALFAC*FSCALFAC)  |
| Pressure at an acoustic point  | Unit force at an excitation point  | 1.0/(LSCALFAC**2)  |
|  | Unit moment at an excitation point | 1.0/(LSCALFAC**3)  |

**FRFCONN**

## FRF Component Explicit Connection for FRF Based Assembly (FBA)

Defines explicit connection data for FRF components in an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2      | 3                     | 4      | 5                     | 6      | 7 | 8 | 9 | 10 |
|---------|--------|-----------------------|--------|-----------------------|--------|---|---|---|----|
| FRFCONN | CONNID | COMPID1/<br>COMPNAME1 | POINT1 | COMPID2/<br>COMPNAME2 | POINT2 |   |   |   |    |

**Examples:**

|         |    |       |     |      |     |  |  |  |  |
|---------|----|-------|-----|------|-----|--|--|--|--|
| FRFCONN | 10 | 5     | 100 | 15   | 200 |  |  |  |  |
| FRFCONN | 20 | STRUT | 25  | WING | 35  |  |  |  |  |

| Descriptor            | Meaning   |
|-----------------------|---|
| CONNID                | Unique identification number of the FRFCONN entry. (Integer > 0)  |
| COMPID <sub>i</sub>   | Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)         |
| COMPNAME <sub>i</sub> | Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed) |
| POINT <sub>i</sub>    | Grid or scalar point identification number. See Remarks 2. through 6. (Integer > 0)   |

**Remarks:**

1. If neither of the FRF components COMPID1/COMPNAME1 and COMPID2/COMPNAME2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINT<sub>i</sub> must be among the connection points of the corresponding FRF component COMPID<sub>i</sub>/COMPNAME<sub>i</sub>. If not, the program terminates the job with a user fatal message.
3. It is, in general, not necessary to have FRFCONN entries when the connections in the FBA process involve only grid points. In the FBA process, all connection grid points of FRF components that have the same basic coordinates are normally connected automatically without any user intervention or specification.
4. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.
5. FRFCONN entries are required if the user wants to specify explicit connections between test FRF component grid points whose coordinates are not available and those of other FRF components. See Remark 10. under the description of the FRFCOMP entry for further details.

6. FRFCONN entries are also required if the user wants to combine scalar points of FRF components in the FBA process.

**FRFFLEX****FRF Component Flexible Connection Specification for FRF Based Assembly (FBA)**

Defines properties for flexible connections between FRF components in an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2                   | 3 | 4                     | 5      | 6                     | 7      | 8                 | 9                 | 10 |
|---------|---------------------|---|-----------------------|--------|-----------------------|--------|-------------------|-------------------|----|
| FRFFLEX | FLEXID              | C | COMPID1/<br>COMPNAME1 | POINT1 | COMPID2/<br>COMPNAME2 | POINT2 | KVALUE/<br>KTABID | BVALUE/<br>BTABID |    |
|         | GEVALUE/<br>GETABID |   |                       |        |                       |        |                   |                   |    |

**Examples:**

|         |      |   |     |    |     |    |       |     |  |
|---------|------|---|-----|----|-----|----|-------|-----|--|
| FRFFLEX | 10   | 1 | 100 | 15 | 200 | 25 | 100.0 | 150 |  |
|         | 0.02 |   |     |    |     |    |       |     |  |

|         |    |   |       |     |      |     |    |      |  |
|---------|----|---|-------|-----|------|-----|----|------|--|
| FRFFLEX | 20 | 4 | STRUT | 120 | WING | 260 | 10 | 0.25 |  |
|---------|----|---|-------|-----|------|-----|----|------|--|

| Descriptor | Meaning   |
|------------|---|
| FLEXID     | Unique identification number of the FRFFLEX entry. (Integer > 0)  |
| C          | A single component number. (Any integer between 1 and 6 for grid points; integer 0 or blank for scalar points.)   |
| COMPIDI    | Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)   |
| COMPNAMEi  | Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)                                   |
| POINTi     | Grid or scalar point identification number. See Remark 2. (Integer > 0)   |
| KVALUE     | Elastic property value (force per unit displacement). See Remark 3. (Real > 0.0 or blank)   |
| KTABID     | Identification number of a TABLEDi entry that defines the elastic property value (force per unit displacement) as a function of frequency. See Remark 3. (Integer > 0 or blank) |
| BVALUE     | Damping property value (force per unit velocity). See Remark 3. (Real > 0.0 or blank)   |
| BTABID     | Identification number of a TABLEDi entry that defines the damping property value (force per unit velocity) as a function of frequency. See Remark 3. (Integer > 0 or blank)     |
| GEVALUE    | Damping coefficient value. See Remarks 4., 5. and 6. (Real > 0.0 or blank)  |
| GETABID    | Identification number of a TABLEDi entry that defines the damping coefficient value as a function of frequency. See Remarks 4., 5. and 6. (Integer > 0 or blank)                |

**Remarks:**

1. If neither of the FRF components COMPID1/COMPNAME1 and COMPID2/COMPNAME2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINTi must be among the connection points of the corresponding FRF component COMPIDi/COMPNAMEi. If not, the program terminates the job with a user fatal message.
3. The KVALUE/KTABID and BVALUE/BTABID fields may not both be blank.
4. The continuation entry is not needed if GEVALUE/GETABID is not to be defined.
5. GEVALUE/GETABID may not be specified unless KVALUE/KTABID is specified.
6. To obtain the damping coefficient, multiply the critical damping ratio  $C/C_0$  by 2.0.
7. It is important to note that this entry by itself does *not* define a connection between the specified points. It merely defines properties for flexible connections between two points whose connection is established either explicitly via an FRFCOMP entry or is implied by automatic connection.
8. The flexible connection properties for component C of connection points POINT1 and POINT2 may be defined on more than one FRFFLEX entry.
9. In the absence of FRFFLEX data, the program assumes rigid connections between the corresponding components.
10. A grid point component may not appear on both an FRFFLEX entry and an FRFRELS entry. If it does, the program identifies such usage and terminates the job with a user fatal message.

**FRFRELS**

## FRF Component Grid Point Release for FRF Based Assembly (FBA)

Defines the degrees-of-freedom of FRF component connection grid points that are not to be connected in an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2   | 3 | 4                     | 5        | 6                     | 7        | 8                     | 9        | 10 |
|---------|-----|---|-----------------------|----------|-----------------------|----------|-----------------------|----------|----|
| FRFRELS | SID | C | COMPID1/<br>COMPNAME1 | GRIDPNT1 | COMPID2/<br>COMPNAME2 | GRIDPNT2 | COMPID3/<br>COMPNAME3 | GRIDPNT3 |    |

**Examples:**

|         |     |    |    |    |      |    |       |    |  |
|---------|-----|----|----|----|------|----|-------|----|--|
| FRFRELS | 100 | 45 | 10 | 15 | BODY | 20 | FRAME | 30 |  |
|---------|-----|----|----|----|------|----|-------|----|--|

|         |    |   |      |    |         |    |  |  |  |
|---------|----|---|------|----|---------|----|--|--|--|
| FRFRELS | 20 | 1 | WING | 25 | NACELLE | 35 |  |  |  |
|---------|----|---|------|----|---------|----|--|--|--|

| Descriptor            | Meaning   |
|-----------------------|---|
| SID                   | Identification number of the FRFRELS entry. (Integer > 0)   |
| C                     | Component number(s). See Remark 1. (Any unique combination of the integers 1 through 6 with no embedded blanks.)                              |
| COMPID <sub>i</sub>   | Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)         |
| COMPNAME <sub>i</sub> | Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed) |
| GRIDPNT <sub>i</sub>  | Grid point identification number. See Remarks 3. and 4. (Integer > 0)   |

**Remarks:**

1. The grid point component(s) specified by C will not be connected in the FBA process.
2. If FRF component COMPID<sub>i</sub>/COMPNAME<sub>i</sub> is not part of the FBA process, then the release data for that FRF component is ignored.
3. GRIDPNT<sub>i</sub> must be among the connection points of the corresponding FRF component COMPID<sub>i</sub>/COMPNAME<sub>i</sub>. If not, the program terminates the job with a user fatal message.
4. If GRIDPNT<sub>i</sub> is not connected to any other grid point in the FBA process, the program ignores the FRFRELS data for this point and issues a warning message indicating this to the user.
5. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.

**FRFSPC1****FRF Component Single-Point Constraint for FRF Based Assembly (FBA)**

Defines single-point constraints for FRF component connection points in an FRF Based Assembly (FBA) process.

**Format:**

| 1       | 2   | 3 | 4                     | 5      | 6                     | 7      | 8                     | 9      | 10 |
|---------|-----|---|-----------------------|--------|-----------------------|--------|-----------------------|--------|----|
| FRFSPC1 | SID | C | COMPID1/<br>COMPNAME1 | POINT1 | COMPID2/<br>COMPNAME2 | POINT2 | COMPID3/<br>COMPNAME3 | POINT3 |    |

**Examples:**

|         |     |   |       |    |         |    |       |    |  |
|---------|-----|---|-------|----|---------|----|-------|----|--|
| FRFSPC1 | 100 | 4 | 10    | 12 | ENGINE  | 23 | FRAME | 31 |  |
| FRFSPC1 | 20  | 1 | STRUT | 25 | NACELLE | 35 |       |    |  |

| Descriptor            | Meaning  |
|-----------------------|--|
| SID                   | Identification number of the single-point constraint set. See Remark 1. (Integer > 0)  |
| C                     | Component number(s). (Any unique combination of the integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.) |
| COMPID <sub>i</sub>   | Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)                    |
| COMPNAME <sub>i</sub> | Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)            |
| POINT <sub>i</sub>    | Grid or scalar point identification number. See Remarks 3. and 4. (Integer > 0)  |

**Remarks:**

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. If FRF component COMPID<sub>i</sub>/COMPNAME<sub>i</sub> is not part of the FBA process, then the single-point constraint data for that FRF component is ignored.
3. POINT<sub>i</sub> must be among the connection points of the corresponding FRF component COMPID<sub>i</sub>/COMPNAME<sub>i</sub>. If not, the program terminates the job with a user fatal message.
4. If POINT<sub>i</sub> is not connected to any other point in the FBA process, the program ignores the FRFSPC1 data for this point and issues a warning message indicating this to the user.
5. Unlike in non-FBA jobs, wherein a degree-of-freedom that has SPCs specified for it yields exact zero results, a degree-of-freedom that has an FRFSPC1 specified for it in an FBA job will, in general, not give exact zero results, but will give results that are nearly zero or very close to zero, usually around 1.0E-12 or so.

**FRFXT**

## Unit Load Degree-of-Freedom Specification for Frequency Response Function (FRF) Computations

Specifies a single degree-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

**Format:**

| 1     | 2     | 3 | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-------|---|-------|---|---|---|---|---|----|
| FRFXT | PNTID | C | LABEL |   |   |   |   |   |    |

**Example:**

|       |    |   |                          |  |  |  |  |  |  |
|-------|----|---|--------------------------|--|--|--|--|--|--|
| FRFXT | 10 | 3 | UNIT LOAD AT LEFT CORNER |  |  |  |  |  |  |
|-------|----|---|--------------------------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PNTID      | Grid or scalar point identification number. (Integer > 0)  |
| C          | A single component number. (Integer 0 or blank for a scalar point; any integer between 1 and 6 for a grid point.)  |
| LABEL      | A string comprising no more than 48 characters (fields 4 through 9) that will be used in the label portion of the FRF output to identify this unit load specification. See Remarks 1. and 2. |

**Remarks:**

1. The small field format must be employed for this entry. If the free field or large field format is employed, the results are unpredictable and in many cases may lead to fatal errors and subsequent termination of the job.
2. The LABEL data must have a non-blank entry in field 4.
3. The FRFXT1 Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
4. Redundant unit load specifications are ignored.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**FRFxit1****Unit Load Degrees-of-Freedom Specification for Frequency Response Function (FRF) Computations**

Specifies degrees-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

**Format:**

| 1       | 2 | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|---|--------|--------|--------|--------|--------|--------|--------|----|
| FRFxit1 | C | PNTID1 | PNTID2 | PNTID3 | PNTID4 | PNTID5 | PNTID6 | PNTID7 |    |

**Example:**

|         |     |    |    |    |    |  |  |  |  |
|---------|-----|----|----|----|----|--|--|--|--|
| FRFxit1 | 123 | 10 | 20 | 30 | 40 |  |  |  |  |
|---------|-----|----|----|----|----|--|--|--|--|

**Alternate Format and Example:**

| 1       | 2   | 3      | 4    | 5      | 6 | 7 | 8 | 9 | 10 |
|---------|-----|--------|------|--------|---|---|---|---|----|
| FRFxit1 | C   | PNTID1 | THRU | PNTID2 |   |   |   |   |    |
| FRFxit1 | 123 | 5      | THRU | 15     |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| C          | Component numbers. (Any unique combination of integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.) |
| PNTIDi     | Grid or scalar point identification numbers. See Remark 1. (Integer > 0)   |

**Remarks:**

1. Points in the THRU range need not all exist.
2. The FRFxit Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
3. Redundant unit load specifications are ignored.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FSICTRL

Defines the analysis type for OpenFSI fluid structure analysis simulations in SOL 400.

**Format:**

| 1       | 2       | 3    | 4    | 5        | 6 | 7 | 8 | 9 | 10 |
|---------|---------|------|------|----------|---|---|---|---|----|
| FSICTRL | SERV_ID | TYPE | FREQ | ANALYSIS |   |   |   |   |    |

**Examples:**

|         |        |          |   |  |  |  |  |  |  |
|---------|--------|----------|---|--|--|--|--|--|--|
| FSICTRL | scafsi | EXPLICIT | 1 |  |  |  |  |  |  |
|---------|--------|----------|---|--|--|--|--|--|--|

| Descriptor | Meaning   | Character   | None     |
|------------|---|-------------|----------|
| SERV_ID    | OpenFSI SCA service name associated with the wetted surface loads (see <a href="#">WETLOAD, 3327</a> ). The OpenFSI SCA service is defined using the CONNECT SERVICE file management section statement. | Character   | None     |
| TYPE       | Type of solution strategy coupling between the external code and SOL 700. TYPE can be either EXPLICIT or IMPLICIT.  | Character   | EXPLICIT |
| FREQ       | External force and displacement update frequency per time step, for the exchange with the external code using the IMPLICIT solution strategy TYPE.  | Integer > 0 | 1        |
| ANALYSIS   | OpenFSI service type, it is either DV or U. The DV indicates the service implements OpenFSI interface, the U means the service implements OpenFSI_Ex interface.   | Character   | DV       |

**Remarks:**

1. This entry is used for nonlinear SOL 400 analysis.
2. In the FMS Section:  
CONNECT SERVICE SERV\_ID 'ExternalCodeVendor.OpenFSI '
3. The SERV\_ID string must be 8 characters or less.

**FSLIST****Free Surface List**

Defines the fluid points (RINGFL entry) that lie on a free surface boundary.

**Format:**

| 1      | 2    | 3      | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|------|--------|------|------|------|------|------|------|----|
| FSLIST | RHO  | IDF1   | IDF2 | IDF3 | IDF4 | IDF5 | IDF6 | IDF7 |    |
|        | IDF8 | -etc.- |      |      |      |      |      |      |    |

**Examples:**

|        |       |      |     |     |     |     |     |     |  |
|--------|-------|------|-----|-----|-----|-----|-----|-----|--|
| FSLIST | 1.0-4 | AXIS | 432 | 325 | 416 | 203 | 256 | 175 |  |
|--------|-------|------|-----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning   |
|------------|---|
| RHO        | Mass density at the surface. (Real > 0.0 or blank; the default is taken from DRHO on the AXIF entry.) |
| IDF1       | Identification number of RINGFL entry. (Integer > 0 or Character = “AXIS” or “LAXIS”) See Remark 5.   |
| IDF2-IDFi  | Identification number of additional RINGFL entries. (Unique Integers > 0)                             |

**Remarks:**

1. This entry is allowed only if an AXIF entry is also present.
2. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. If the fluid density varies along the boundary, there must be one FSLIST entry for each interval between fluid points.
5. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS. See Remark 2.

**FTGDEF****Fatigue Element Definitions**

Defines elements and their associated fatigue properties to be considered for fatigue analysis for time domain SOLutions 101, 103, 112 and frequency domain SOLutions 108 and 111.

Format (SOL 101/103/112):

| 1      | 2        | 3       | 4       | 5       | 6       | 7       | 8       | 9       | 10 |
|--------|----------|---------|---------|---------|---------|---------|---------|---------|----|
| FTGDEF | ID       | TOPSTR  | PFTGID  | TOPDMG  | NENTS   | maxENTS |         |         |    |
|        | "ELSET"  | ELSID1  | PFTGID1 | ELSID2  | PFTGID2 | ELSID3  | PFTGID3 |         |    |
|        |          | ELSID4  | PFTGID4 | ...     | ...     | ...     | ...     |         |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |
|        | "SPOTW"  | ELSID1  | PFTGID1 | ELSID2  | PFTGID2 | ELSID3  | PFTGID3 |         |    |
|        |          | ELSID4  | PFTGID4 | ...     | ...     | ...     | ...     |         |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |
|        | "SEAMW"  | ELSID1  | PFTGID1 | NDSID1  | WELD1   | TYPE1   |         |         |    |
|        |          | ELSID2  | PFTGID2 | NDSID2  | WELD2   | TYPE2   |         |         |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |
|        | "XELSET" | XELSID1 | XELSID2 | XELSID3 | XELSID4 | XELSID5 | XELSID6 | XELSID7 |    |
|        |          | XELSID8 | ...     | ...     | ...     | ...     | ...     | ...     |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |

Format (SOL 108/112):

| 1      | 2        | 3       | 4       | 5       | 6       | 7       | 8       | 9       | 10 |
|--------|----------|---------|---------|---------|---------|---------|---------|---------|----|
| FTGDEF | ID       | TOPRMS  |         |         |         |         |         |         |    |
|        | "ELSET"  | ELSID1  |         | ELSID2  |         | ELSID3  |         |         |    |
|        |          | ELSID4  |         | ...     |         | ...     |         |         |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |
|        | "XELSET" | XELSID1 | XELSID2 | XELSID3 | XELSID4 | XELSID5 | XELSID6 | XELSID7 |    |
|        |          | XELSID8 | ...     | ...     | ...     | ...     | ...     | ...     |    |
|        |          | -etc.-  |         |         |         |         |         |         |    |

Examples:

|        |       |       |   |    |   |  |  |  |  |
|--------|-------|-------|---|----|---|--|--|--|--|
| FTGDEF | 22    | 100.0 | 3 |    |   |  |  |  |  |
| FTGDEF | 22    |       |   |    |   |  |  |  |  |
|        | ELSET | 14    | 3 | 15 | 4 |  |  |  |  |

|        |       |    |   |    |   |    |  |  |  |
|--------|-------|----|---|----|---|----|--|--|--|
| FTGDEF | 22    |    |   |    |   |    |  |  |  |
|        | SPOTW | 44 | 1 | 45 | 2 | 46 |  |  |  |

|        |       |    |    |    |        |      |  |  |  |
|--------|-------|----|----|----|--------|------|--|--|--|
| FTGDEF | 22    |    |    |    |        |      |  |  |  |
|        | SEAMW | 44 | 99 | 31 | FILLET | TOE  |  |  |  |
|        |       | 45 | 99 |    | LASER  | ROOT |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number called out by the FATIGUE case control. (Integer > 0).  |
| TOPSTR     | <b>SOL 101, 103, 112:</b> Top stress percentage. Only entities with combined stress in this top percentage are retained and report results. Should not be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank.   |
| TOPRMS     | <b>SOL 108, 111:</b> Top RMS (root mean square) stress percentage. Only entities with RMS stress levels in this top percentage are retained and report results.<br><br>(0.0 < Real ≤ 100.0; Default = blank - 100% will be used) See Remark 8.   |
| PFTGID     | <b>SOL 101, 103, 112</b> only. ID of a PFTG entry for associating fatigue properties to all elements of the model. Ignored if ELSET flag is present and should be left blank in this case. (Optional, Integer > 0). See Remark 2.  |
| TOPDMG     | <b>SOL 101, 103, 112</b> only. Top damage percentage. Only elements with damage in this top percentage will be retained and report results. (0.0 < Real ≤ 100.0; Default = blank). Should not be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank. See Remark 8.  |
| NENTS      | <b>SOL 101, 103, 112</b> only. Specifies the number of actual entities to output in order to limit or further reduce output file sizes based on most damage or maximum stress/strain range. A positive value indicates the criterion for entity filtering is based on the most damaged entities. A negative number indicates the filtering criteria is based on maximum stress/strain range. (Integer != 0; Default = blank). Not valid for SOL 200; leave blank. See Remark 10. |
| maxENTS    | <b>SOL 101, 103, 112</b> only. This is used only if STROUT=4 is specified in the FATIGUE case control. If the number of entities requested is greater than maxENTS, the job stops to avoid performance issues. Setting this to a large number can severely affect performance. When using STROUT=4, the number of entities should be limited. (Integer > 0; Default = 100).  |
| ELSET      | Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for consideration in the fatigue analysis. (Optional, Character = ELSET) See Remark 9.  |
| ELSIDi     | ID of a SET1, SET3, or SET4 entry listing entities of the model (elements) to be included in the fatigue analysis. (Integer > 0). See Remark 3.  |
| PFTGIDI    | ID of a PFTG entry, which indicates the fatigue property associated to the preceding entities defined by ELSIDi. (Optional, Integer > 0). See Remark 2.  |

| Descriptor      | Meaning   |
|-----------------|---|
| SPOTW           | <b>SOL 101, 103, 112</b> only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of spot welds. See Remark 5. and 6.  |
| ELSIDi          | Same as ELSIDi under ELSET above. See Remark 3.   |
| PFTGIDi         | Same as PFTGIDi under ELSET above.  |
| SEAMW           | <b>SOL 101, 103, 112</b> only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of seam welds. See Remark 5.   |
| ELSIDi          | Same as ELSIDi under ELSET above. These are to elements that make up the seam weld toe, root, or throat. See Remark 3.  |
| PFTGIDi         | Same as PFTGIDi under ELSET above.  |
| NDSIDi          | ID of a SET1 or SET3 entry listing grids of the elements defined by ELSIDi to be retained in the analysis. These grids define the seam line of the seam weld. If left blank, all nodes of the elements are retained. Nodes defined that are not part of ELSIDi are ignored. (Optional, Integer >0). |
| WELD <i>i</i>   | Seam weld definition. One of the following: FILLET, OVERLAP, LASER, EDGE, or GENERIC, which define either a fillet, overlap, laser overlap, laser edge overlap or generic seam weld, respectively. (Character; Default=GENERIC). See Remark 7.  |
| TYPE <i>i</i>   | The type location on the seam weld that this set of elements represent. One of the following: TOE, ROOT, or THROAT. (Character; Default = TOE for all but WELD <i>i</i> = LASER where Default = ROOT). See Remark 7.  |
| XELSET          | Flag indicating that sets of elements to be excluded from the fatigue analysis will follow. (Optional, Character). See Remark 4.  |
| XELSID <i>i</i> | ID of a SET1 or SET3 entry listing elements of the model to be excluded from the fatigue analysis. (Integer>0). See Remark 3.   |

#### Remarks:

1. **FTGDEF** bulk data entries are ignored if not selected by a **FATIGUE** case control. If no **FTGDEF** is present for a given fatigue analysis, all elements of the model that have fatigue material properties defined will be used with default properties except for fatigue analysis of spot and seam welds. See Remark 5. below.
2. If no PFTGID or PTFGIDi is specified, default properties will be assigned to the entities.
3. If a SET3 is specified, field 3 of the SET3 entry must be set to "ELEM". The SET4 entry must be specified to select elements by property ID. The following elements referenced by PSHELL, PSHEAR, and PSOLID properties are supported for standard S-N and  $\epsilon$ -N fatigue analysis: CQUAD4, CQUADR, CQUAD8, CSHEAR, CTRIA3, CTRIAR, CTRIA6, CHEXA, CPENTA,

and CTETRA. The following elements referenced by PBAR, PBEAM, PSOLID, and PWELD properties are supported for fatigue analysis of spot welds: CBAR, CBEAM, CHEXA, CWELD. Only elements referenced by PSHELL properties are available for fatigue analysis of seam welds, excluding TRIA3 as no corner stresses are available from this element.

4. If only the XELSET flag is present, then the entire model is included in the fatigue analysis less the excluded entities.
5. For fatigue analysis of spot or seam welds the FTGDEF card is required with the corresponding keyword and line defining the elements of interest.
6. Elements used to represent spot welds are typically very stiff bar or beam (CBAR/CBEAM) elements, weld (CWELD) elements, or individual solid (CHEXA) elements. These elements connect the two metal sheets defined by shell elements (CSHELL). Bars and beams must connect directly to the grids of the shells, whereas welds and fasteners only need to pierce the shell elements. If individual solid elements are used, face G1-G2-G3-G4 must have its grids connected to shell elements that define the top sheet via RBE3 rigid elements. And face G5-G6-G7-G8 must have corresponding RBE3 elements connecting the bottom sheet. Some CWELD options auto-generate grids on the top and bottom metal sheets (when only GS grid is defined on CWELD entry with no GA/GB grids); the fatigue results will be associated to these generated grids.
7. The normals of the throat elements should point outward toward the welder, except for laser overlap, in which case the normals just need to be consistently the same direction. The elements defining the toe and root of the weld must have the top of the shell (Z2 layer) be the side where the crack is expected to develop. For full descriptions of the throat, root, and toe elements for the various seam welds, please see the *MSC Nastran Embedded Fatigue User's Guide*, which show proper modeling techniques. The WELDi and TYPEi entries are used for labeling purposes only and have no effect on internal calculations.
8. Use of TOPSTR/TOPRMS and/or TOPDMG in the same analysis allows the user to perform a 2-pass or a 3-pass run to speed up the analysis times of very large models and more quickly determine the critical damage locations.
  - If TOPSTR/TOPRMS is used alone (TOPDMG is left unspecified), than a 2-pass analysis is performed where the first (1st) pass determines the elements with the highest combined stresses based on the stress combination (COMB field on [FTGPARM](#) entry) and the largest range of the combined loading time histories. The second (2nd) pass computes damage on only the remaining elements.
  - If TOPDMG is used alone (TOPSTR=100% or left unspecified), then a 2-pass analysis is performed where the first (1st) pass determines the elements with the highest damage based on a set of reduced loading time histories. The second (2nd) pass computes damage on only the remaining elements using the fully populated time history loading.
  - If TOPSTR and TOPDMG are both specified, then a 3-pass analysis is performed where the first (1st) pass eliminates all but the highest stressed elements and the second (2nd) pass eliminates more elements based on damage due to a reduced set of time histories. The third (3rd) pass computes damage on the remaining element with the fully populated time history loading.

- If neither TOPSTR/TOPRMS or TOPDMG are specified (both are blank), then a 1-pass analysis is done on all the specified elements with fully populated loading time histories. This is the default scenario.

When using TOPDMG, the method in which the reduced time histories are created is determined by the settings on the RAINFLOW line of the [FTGPARM](#) entry. If this entry is absent or the RAINFLOW line is not present, then the default is RTYPE=LOAD with PCTRED=50%.

9. Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses through the FTGPARM entry (SRESOLVE field = YES). Otherwise a fatal condition is flagged and the analysis stops.
10. This parameter can be used by itself or in conjunctions with TOPSTR and/or TOPDMG. Only entities that pass this filter are reported in the f06 file and/or output to the OUTPUT2 and/or HDF5 files. If FATIGUE(STROUT=4) case control is used it is highly recommended that NENTS be used to limit the output. Temporary CSV files are created for each entity and F06 output can be enormous if not filtered using NENTS. If NENTS is used with the SRESOLVE option on the [FTGPARM](#) entry, then the number of entities printed may be more as the SRESOLVE option saves the number of grids and each element associated with those grids.

**FTGEVNT****Fatigue Loading Events**

Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random vibration fatigue using SOL 108 or SOL 111 by referencing FTGLOAD entries.

**Format (SOL 101, 103, 108, 111):**

| 1       | 2      | 3      | 4      | 5      | 6       | 7      | 8      | 9      | 10 |
|---------|--------|--------|--------|--------|---------|--------|--------|--------|----|
| FTGEVNT | ID     | FLOAD1 | FLOAD2 | FLOAD3 | FLOAD4  | FLOAD5 | FLOAD6 | FLOAD7 |    |
|         |        | FLOAD8 | ...    | ...    | ...     | ...    | ...    | ...    |    |
|         |        | -etc-  |        |        |         |        |        |        |    |
|         | "NAME" |        |        |        | EVNTNAM |        |        |        |    |

**Example:**

|         |      |   |    |  |  |  |  |  |            |
|---------|------|---|----|--|--|--|--|--|------------|
| FTGEVNT | 22   | 4 | 11 |  |  |  |  |  |            |
|         | NAME |   |    |  |  |  |  |  | rough_road |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique ID. See Remark 1.   |
| FLOADi     | ID of a FTGLOAD entry (Integer > 0, no default). See Remark 5.   |
| NAME       | Flag indicating that a name is to be associated with this load event. See Remark 2.  |
| EVNTNAM    | Event name associated with this event (Character). Can span fields 3 through 9, however for practical purposes, the name is truncated in f06 output. No spaces are allowed. See Remark 3. and 4. |

**Remarks:**

1. Each **FTGEVNT** ID must be unique relative to all other **FTGEVNT** and **FTGSEQ** IDs.
2. Event names are passed to the f06 print file and displayed as the name of the event as opposed to the event ID. If no name is given, the event is referred to by its ID.
3. EVNTNAM cannot have numbers at the start of any field. For example, using 8 character formatted fields, the first character of fields 3-9 cannot be numeric (columns 1, 9, 17, etc. of the EVNTNAM field). These would be acceptable:  
"Flights" or "F\_l\_i\_g\_h\_t\_s".  
But these would not be acceptable:  
"1\_Flights" or "F\_l\_i\_g\_2ts"  
because the "1" and the "2" are in columns 1 and 9 of the EVNTNAM field.

4. EVNTNAM cannot handle spaces within its field as one might expect. If you wish to use spaces, make sure that once a space is used the next character does not start until the start of the next field. For example, using 8 character formatted fields, the following would be acceptable:  
“My\_flights” or “My        Flights”.  
But this would not be acceptable:  
“My Flights”  
and would end up just being displayed as “Myts” because once a space is encountered within the 8 character field, the code interprets that as no more character data until it processes the next 8 character field. It is best not to use spaces in EVNTNAM.
5. For random vibration fatigue (**SOL 108 and 111**),
  - Only one FTGLOAD of TYPE=PSD can be referenced
  - Only one FTGLOAD of TYPE=STATIC can be referenced
  - A FTGEVNT referencing a FTGLOAD of TYPE=STATIC by itself is not allowed
  - A FTGEVNT referencing a FTGLOAD of TYPE=SINE or NARROW by itself (or themselves) is allowed for pure deterministic and/or sine sweeps
  - Only single load PSD random input allows mixed deterministic, sine sweep, and static offsets on random loading. Only a static offset (FTGLOAD of TYPE=STATIC) can be mixed with multi-load PSD random input.
  - FTGSEQ entry referencing FTGEVNT entries cannot mix events containing single load and multi-load PSDs.
  - All multi-load PSD events must all have the same number of channels (input loads).
  - When only FTGLOAD entries of TYPE=SINE or NARROW are referenced (i.e., only deterministic loading with no random loading input), it is recommended that the ATYPE field of the FTGPARM entry be set to SINE.

**FTGLOAD****Fatigue Cyclic Loading Variation**

Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random PSD, deterministic, sine sweep and/or static offset loading for SOL 108 and SOL 111.

Format (SOL 101 and 103):

| 1       | 2       | 3     | 4      | 5   | 6             | 7              | 8    | 9    | 10 |
|---------|---------|-------|--------|-----|---------------|----------------|------|------|----|
| FTGLOAD | ID      | TID   | LCID   | LDM | SCALE/<br>MAX | OFFSET/<br>MIN | TYPE | CHNL |    |
|         | "UNITS" | EQUIV | EQNAME |     |               |                |      |      |    |

Format (SOL 108 and 111):

| 1       | 2         | 3      | 4     | 5      | 6      | 7       | 8      | 9 | 10 |
|---------|-----------|--------|-------|--------|--------|---------|--------|---|----|
| FTGLOAD | ID        | TID    | LCID  |        | SCALE  | OFFSET  | TYPE   |   |    |
|         | "DETLOAD" | F1     | G1/H1 | B1     | F2     | G2/H2   | B2     |   |    |
|         |           | F3     | G3/H3 | B3     | ...    | ...     | ...    |   |    |
|         |           | -etc.- |       |        |        |         |        |   |    |
|         | "SWEEP"   | F1     | G1/H1 |        | Fn     | Gn/Hn   |        |   |    |
|         |           | SWTYPE | SWNUM | SWRATE | NSWEEP | FRFUNIT | LDUNIT |   |    |

Examples:

|         |       |     |         |     |     |     |    |  |  |
|---------|-------|-----|---------|-----|-----|-----|----|--|--|
| FTGLOAD | 55    | 4   | 2       | 1.0 | 1.0 | 0.0 |    |  |  |
| FTGLOAD | 23    | 4   | 5       | 1.0 | 1.0 | 0.0 | DB |  |  |
|         | UNITS | 5.5 | Flights |     |     |     |    |  |  |

| Descriptor    | Meaning  |
|---------------|--|
| ID            | Unique ID which is referenced by a FTGEVNT entry or directly by a FATIGUE case control in the case of SOL 101 only for a single load (Integer > 0). See Remark 1.  |
| TID           | <ul style="list-style-type: none"> <li>■ <b>SOL 101, 103:</b> Table ID of a TABLFTG (or TABLED1) entry that defines the time variation of the loading or the ID of a UDNAME entry for external definition of the loading time variation (Integer &gt; 0). See Remarks 2. through 6. Ignored if TYPE=CONST or STATIC.</li> <li>■ <b>SOL 108, 111:</b> leave blank unless:           <ul style="list-style-type: none"> <li>■ TYPE=STATIC: UDNAME ID referencing an external OP2 file containing stress results for static loading offsets. Leave blank (and set LCID = 0 if a constant offset defined solely by the OFFSET field is desired).</li> <li>■ TYPE = PSD with TIM2PSD: UDNAME ID referencing an external RPC or CSV file when time history to PSD conversion is requested. A TIM2PSD entry must be present in this case, otherwise TID is ignored.</li> </ul> </li> </ul>  |
| LCID          | <ul style="list-style-type: none"> <li>■ <b>SOL 101:</b> Subcase ID of a linear static solution SUBCASE (Integer &gt; 0, no default).</li> <li>■ <b>SOL 103:</b> Mode number for SOL 103 (Integer &gt; 0, no default).</li> <li>■ <b>SOL 108, 111:</b> <ul style="list-style-type: none"> <li>■ TYPE = PSD with no TIM2PSD: SID of RANDPS entry(ies). TID should be left blank in this case.</li> <li>■ TYPE = PSD with TIM2PSD: -1 for fully correlated multi-input loading, -2 for uncorrelated multi-input loading.</li> <li>■ TYPE = STATIC: SUBCASE ID of a static SUBCASE (freq=0.0). A TID must also be defined in this case to point to an external OP2 file. Set to zero (0) if a constant offset defined solely by the OFFSET field is desired. TID should be left blank in this case.</li> <li>■ TYPE=SINE or NARROW: SUBCASE ID of transfer function (TF) used for deterministic or sweep loading. TID should be left blank in this case.</li> </ul> </li> </ul> |
| LDM           | <b>SOL 101, 103 only.</b> Largest magnitude of the applied load (in the same units used to define the load time variation in field TID) used to normalize the load (Real, default=1.0). This effectively scales the stress to simulate a stress state due to a unit load. See Remark 7.  |
| SCALE/<br>MAX | <p><b>SOL 101, 103:</b> Scale factor applied to the load time history (Real, default=1.0). See Remark 7. Or the 1st peak level of a constant amplitude signal if TYPE=CONST (Real, default=1.0).</p> <p><b>SOL 108, 111:</b> Acts as an additional scale factor on the input PSD. This applies only to TYPE=PSD (Real, default=1.0).</p>   |

| Descriptor     | Meaning  |
|----------------|--|
| OFFSET/<br>MIN | <b>SOL 101, 103:</b> Offset applied to the load time history (Real, default=0.0). See Remark 7. Or the 2nd peak level of a constant amplitude signal if TYPE=CONST (Real, default = -1.0). MAX can be < or > than MIN for TYPE=CONST.<br><br><b>SOL 108, 111:</b> Offset of resultant stress PSD in stress units. This has the effect of adding a mean stress effect to the random load response. This only applies to TYPE=STATIC and if TID is blank, this is only a constant offset. With TID a variable offset plus this constant offset is applied (Real, default = 1.0).   |
| TYPE           | Flag indicating the type of load being defined. Values can be: blank, "DB", "DAC", "RPC", "CONST", "PSD", "SINE", "NARROW" or "STATIC". Default is blank. See Remarks 2. through 6.  |
| CHNL           | <b>SOL 101 and 103 only.</b> Channel of referenced RPC file (TYPE=RPC). (Integer >0, Default is blank). See Remark 12.   |
| "UNITS"        | <b>SOL 101 and 103 only.</b> Flag indicating that a fatigue equivalent unit name is defined for this loading. See Remarks 9. and 10.   |
| EQUIV          | Number of equivalent units (Real>0.0, default=1.0). See Remarks 9. and 10.   |
| EQNAME         | Equivalent name of this loading history. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remarks 9. and 10.   |
| "DETLOAD"      | <b>SOL 108 and 111 only.</b> Flag indicating that deterministic load input is to follow for TYPE=SINE or NARROW.<br><br>Fi Frequency of sine wave for TYPE=SINE or center frequency of narrow band frequency block for TYPE=NARROW. (Real>0.0; no Default).<br><br>Gi Height of sine wave (Gi) in units of peak amplitude (FE stress units) for TYPE=SINE.<br>Hi Height of narrow band PSD block (Hi) in units of (stress) <sup>2</sup> /Hz for TYPE=NARROW. (Real>=0.0; no Default).<br><br>Bi Width of narrow band frequency block. Only supplied for TYPE=NARROW. (Real>=0.0; no Default).  |
| "SWEEP"        | <b>SOL 108 and 111 only.</b> Flag indicating that sine sweep load input is to follow for TYPE=SINE (not supported for NARROW).<br><br>F1/Fn F1/Fn Frequency of 1st and last sine waves for TYPE=SINE. (Real>0.0; no Default).<br>G1/H1 Height of 1st (G1) and last (Gn) sine waves in units of peak amplitude (FE stress units) for TYPE=SINE. (Real>=0.0; no Default).<br>Gn/Hn Gn/Hn Height of 1st (G1) and last (Gn) sine waves in units of peak amplitude (FE stress units) for TYPE=SINE. (Real>=0.0; no Default).<br>SWTYPE The sweep type: Decibel (DB), Octave (OCT), or Linear (HZ). (Character; Default = HZ)<br>SWNUM Number of sine/narrow bands to use (Integer>0 where N=SWNUM+1; Default = 50)<br>SWRATE Sweep rate per second in HZ (default), DB or OCT. (Real>0.0; No Default)<br>NSWEEP Number of sweep passes (Integer>0; No Default). |

| Descriptor | Meaning  |
|------------|--|
| FRFUNIT    | Units of loading used to create FRF - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A)   |
| LDUNIT     | Units of loading used to define sweep - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A) |

**Remarks:**

1. If a FATIGUE case control command invokes an ID that is present on both an FTGSEQ entry and a FTGLOAD entry, the FTGSEQ request will be honored and the FTGLOAD entry will be ignored unless it is referenced on a FTGEVNT entry.
2. For SOL 101 and 103, if the TYPE field is blank then TID references a TABLFTG or TABLED1 entry. For SOL 108 or 111, TYPE must be supplied.
3. For SOL 101 and 103, if TYPE=DB, DAC or RPC, then TID references a UDNAME ID. A UDNAME entry must be supplied in this case to specify the file and path of the externally defined load time variation. This file is expected to be in standard DAC file format or channel data in the form of a RPC file. TYPE=DB is the same as TYPE=DAC. You cannot mix TYPE=RPC designations with any other TYPES within the same fatigue analysis (FATIGUE=FID). If RPC is used, all FTGLOAD entries must use TYPE=RPC. The same RPC file must be referenced via UDNAME for all FTGLOADs of the same event (FTGEVNT). For SOL 108 and 111, if TYPE=STATIC or if TYPE=PSD with TIM2PSD entry active, the UDNAME references an external OP2 file or RPC/CSV channel data file, respectively.
4. For SOL 101 and 103, if TYPE is STATIC, the TID field should be left blank as it will be ignored. STATIC indicates that the stress state from the specified LCID is to act as a static offset with no load time variation when performing the linear superposition, which will give every element a different offset defined by the stress state at each element of the specified subcase, as opposed to simply specifying the OFFSET field, which gives every element the same offset. If the "STATIC" flag is specified, there must be at least two FTGLOAD entries defined and called out by a FTGEVNT entry, one of which must be time varying (see Remark 7.). For SOL 108 and 111, if TYPE is STATIC, the TID field must be supplied.
5. For SOL 101 and 103, if TYPE is CONST, the TID, LDM, and CHNL fields are ignored. The MAX and MIN fields are used to define the maximum and minimum values of a constant amplitude cycle. TYPE=CONST cannot be mixed with any other TYPE values for a given fatigue analysis (FATIGUE=FID). All loading must be of TYPE=CONST. This is sometimes referred to as block loading.
6. For modal analysis using SOL 103, the referenced load variation defines the modal participation factors for the referenced mode.
7. For SOL 101 and 103, the LDM, SCALE, and OFFSET are used together in the following manner to scale/modify the stress state in order to determine the resulting stress time variation:

$$\sigma_{ij}(t) = \left( P(t) \times \frac{\sigma_{ij,l}}{LDM} \times \text{SCALE} \right) + \text{OFFSET}$$

8. where  $\sigma_{ij}(t)$  is the resulting stress tensor at time t,  $\sigma_{ij,l}$  is the stress tensor from the subcase or mode defined by the LCID field, and P(t) is the y value of the load-time history at time t as defined by the TID field. For multiple loads, the principle of linear superposition is used to combine all loads for a single event. For SOL 108 and 111, LDM and SCALE act as a divisor and multiplier, respectively, to the defined loading. OFFSET is only used for TYPE=STATIC and acts as an additional constant offset over all entities.
9. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the stress time history. If this FTGLOAD is referenced by a FTGEVNT, then the equivalent units on this entry are ignored and those on the FTGSEQ entry take precedent. Only if a FTGLOAD is directly referenced by a FATIGUE case control are the fatigue equivalent units used as defined on the FTGLOAD entry.
10. Example of using equivalent units: If one repeat of the defined time history is equivalent to 5 times around a test track, the equivalent unit name, EQNAME, might be "*laps*," and the equivalent unit, EQUIV, would be 5.0. Fatigue life will be reported in these units if defined, otherwise they are reported as *Repeats* of the loading. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
11. All FTGLOAD entries referenced by a FTGEVNT should reference different SUBCASEs for SOL 101 (or modes for SOL 103) and must have time variations consisting of the same number of points.
12. CHNL is only used for TYPE=RPC for SOL 101 and 103. If supplied, the specified channel of the referenced RPC file is used. If it is left blank, the next available channel sequentially from the last one referenced will be used. For example, if there are three FTGLOAD entries for a specific event and CHNL is blank for all three, the 1st one will use channel 1, the 2nd one will use channel 2 and the 3rd will use channel 3. If in this example the 1st specifies CHNL=11 and the others are blank, then the channels used will be 11, 12, and 13. If the 1st is left blank and the 2nd references CHNL=12, then the channels used will be 1, 12, and 13.

**FTGPARM****Fatigue Parameters**

Defines parameters for a fatigue analysis in time domain SOLs 101, 103, and 112 and in frequency domain SOLs 108 and 111.

**Format (SOLs 101, 103, 112):**

| 1       | 2                          | 3      | 4      | 5       | 6       | 7       | 8       | 9        | 10 |
|---------|----------------------------|--------|--------|---------|---------|---------|---------|----------|----|
| FTGPARM | ID                         | TYPE   | FACTOR | NTHRD   | LOGLVL  | LAYER   | STROUT  |          |    |
|         | "STRESS"<br>or<br>"STRAIN" | COMB   | CORR   | PLAST   | LOC     | INTERP  | RECOVER | SRESOLVE |    |
|         |                            | NANGLE |        |         |         |         |         |          |    |
|         | "RAINFLOW"                 | RTYPE  | GATE   | PCTRED  |         |         |         |          |    |
|         | "CERTNTY"                  | SURV   |        |         |         |         |         |          |    |
|         | "FOS"                      | OPTION | LIFE   | BACKACC | MAXFAC  | MINFAC  |         |          |    |
|         | "DAMAGE"                   | CHECK  | FLOOR  | MAXDAM  | FAILDAM |         |         |          |    |
|         | "SPOTW"                    | COMB   | CORR   | NANGLE  | SWLOC   | MIDDLE  | TORSION |          |    |
|         | "SEAMW"                    | COMB   | CORR   | THICK   | LOCSM   | RESENT  |         |          |    |
|         | "MULTI"                    | MMTHD  | NONLWR | NONUPR  | BIAXLWR | BIAXMID | BIAXUPR | ZERO     |    |
|         |                            | GATE   |        |         |         |         |         |          |    |
|         | "NAVG"                     | MTHD   | OUTPUT | NORMAL  |         |         |         |          |    |

**Format (SOLs 108, 111):**

| 1       | 2        | 3     | 4      | 5       | 6       | 7       | 8       | 9     | 10 |
|---------|----------|-------|--------|---------|---------|---------|---------|-------|----|
| FTGPARM | ID       | TYPE  |        |         | LOGLVL  | LAYER   |         |       |    |
|         | "STRESS" | COMB  | CORR   | PLAST   | LOC     |         | RECOVER |       |    |
|         | "FOS"    |       | LIFE   |         |         |         |         |       |    |
|         | "VIBFTG" | ATYPE | MAXSTR | CLIPLVL | MAXPEAK | STRBINS | MAXFREQ | NCALC |    |

**Examples:**

|         |    |    |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|
| FTGPARM | 22 | EN |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|

|         |        |       |      |  |      |  |       |  |  |
|---------|--------|-------|------|--|------|--|-------|--|--|
| FTGPARM | 22     | SN    |      |  |      |  |       |  |  |
|         | STRESS | SGVON | NONE |  | NODA |  | CUBIC |  |  |

|         |       |    |     |   |  |  |  |  |  |
|---------|-------|----|-----|---|--|--|--|--|--|
| FTGPARM | 22    | SN |     |   |  |  |  |  |  |
|         | SEAMW |    | FKM | 0 |  |  |  |  |  |

|         |       |        |        |    |   |   |   |  |  |
|---------|-------|--------|--------|----|---|---|---|--|--|
| FTGPARM | 22    | SN     |        |    |   |   |   |  |  |
|         | SPOTW | STNDRD | SIMPLE | 18 | 3 | 0 | 0 |  |  |

|         |       |      |      |     |      |      |     |      |  |
|---------|-------|------|------|-----|------|------|-----|------|--|
| FTGPARM | 22    | EN   |      |     |      |      |     |      |  |
|         | MULTI | AUTO | 0.25 | 0.5 | -0.6 | 0.25 | 0.6 | 10.0 |  |
|         |       | 20.0 |      |     |      |      |     |      |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique ID of the FTGPARM entry called out by a FATIGUE case control (Integer > 0). See Remark 1.  |
| TYPE       | <b>SOL 101, 103, 112:</b> Type of fatigue analysis: "SN" or "EN" (Character, Default=SN).   |
|            | <b>SOL 108, 111:</b> Type of fatigue analysis: "SN" or "EN" (Character, Default=SN). This is used only when there is a conflict with the defined material properties, i.e., both S-N and ε-N properties exist for the same material MID. Otherwise it is ignored and the material properties define the type of fatigue analysis to be performed..  |
| FACTOR     | <b>SOL 101, 103, 112</b> only. Global scale factor to be applied to combined resultant stress output (Real>0.0, Default=1.0).   |
| NTHRD      | <b>SOL 101, 103, 112</b> only. Number of threads to use for parallel processing for this fatigue analysis (Integer $\geq 0$ ; Default = 1). Zero (0) is used as a flag to tell the code to automatically determine the number of threads to use.  |
| LOGLVL     | <b>SOL 101, 103, 112:</b> Level of messaging sent to the log file (Integer $\geq 0$ , Default = 0; 0=None, 1=Error, 2=Info, 3=Low, 4=Medium, 5=High). Note that LOGLVL >3 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest.   |
|            | <b>SOL 108, 111:</b> Level of output to generate (Integer $\geq 0$ , Default = 0; 0=Standard, 1=Output all PSD responses, 2=Additionally output all Rainflow data). Note that LOGLVL >0 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest only. See Remark 16.   |
| LAYER      | For shell elements, the output results layer to print to the f06 file. Values can be 0=Worst, 1=Top(Z2), 2=Bottom(Z1). (Integer, Default = 0). This is for printed output only. The analysis produces results for both layers, which are always available through the MASTER/DBALL, Output2, or other files for graphical postprocessors. See the <a href="#">FATIGUE</a> case control entry.   |
| STROUT     | <b>SOL 101, 103, 112</b> only. Requests output of the actual stresses (or strains) used in standard SN and eN fatigue analyses. Values can be 0=No Output, 1=Print Output, 2=Plot Output. (Integer, Default = 0). Printed output is placed in the F06 file and also available in an OESFTG data block for postprocessing purposes. Plot only places the data in the OESFTG data block. This request is overridden by the <a href="#">FATIGUE</a> case control if it also includes a STROUT request. |

| Descriptor  | Meaning  |
|-------------|--|
| "STRESS" or | Flag indicating that stress is used in the fatigue calculation. See Remark 2.  |
| "STRAIN"    | SOL 101, 103, 112 only. Flag indicating that strain is used in the fatigue calculation. Not valid for TYPE = SN. See Remark 2.   |
| COMB        | Stress/strain combination to use in the fatigue analysis. Acceptable values are listed in <a href="#">Table 9-17</a> after the Remarks below (Character; Default= ABSMAXPR).   |
| CORR        | Mean stress correction to use in the fatigue analysis. Acceptable values are listed in <a href="#">Table 9-18</a> and <a href="#">Table 9-19</a> after the Remarks below (Character; Default=None).  |
| PLAST       | SOL 101, 103, 112: Plasticity correction for TYPE = EN. Value can be "NEUBER," or "SEEGER" for Neuber or Hoffmann-Seeger methods, respectively (Character; Default=NEUBER). See Remark 3.<br><br>SOL 108, 111: Only NEUBER correction is available.  |
| LOC         | SOL 101, 103, 112: Location to report fatigue lives. Valid values are "NODA", "NODE", or "ELEM" based on nodal averaged, element nodal, or element centroid stresses or strains, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4. Also see the "NAVG" field and parameters.<br><br>SOL 108, 111: Location to report fatigue lives. Valid values are "NODE", "NODA", or "ELEM" based on element nodal, nodal averaged, or element center stresses, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4. |
| INTERP      | SOL 101, 103, 112 only. Interpolation limit for multi-curve mean stress correction method (Integer $\geq 0$ ; 0=Use Max Curve, 1=Extrapolate; Default = 0).  |
| RECOVER     | Stress recovery method to determine stresses/strains for the fatigue calculation. Valid values are "SGAGE", "CORNER", "BILIN", "CUBIC" or "CENTER" (Character; Default = CORNER if LOC=NODA or NODE; Default = CENTER if LOC=ELEM and any other setting is ignored). These are the same values used for the STRESS (or STRAIN) case control output request. See LOC field and Remark 4.  |
| SRESOLVE    | SOL 101, 103, 112 only. Request for surface resolved, 2D stress state. Used only for 3D solid elements. (Character YES or NO; Default = NO). See Remark 5.   |

| Descriptor | Meaning   |
|------------|---|
| NANGLE     | The number of calculation angles for 2D critical plane analysis (COMB=CRITICAL) ( $1 \leq \text{Integers} \leq 360$ ; Default = 36, i.e. every 10 degrees). Practical values are 360, 180, 120, 90, 72, 60, 45, 40, 36, 30, 24, 18, 15, 12, 10, 9, 8, 6, 5, 4 for every 1, 2, 3, 4, 5, 6, 8, 9, 10, 12, 15, 20, 24, 30, 36, 40, 45, 60, 72, 90 degrees, respectively. The more angles computed, the more compute intensive the analysis will be. In actuality, only half the angles are computed since the principals are the same 180 degrees opposite any computed angle. |
| "RAINFLOW" | <b>SOL 101, 103, 112</b> only. Flag indicating that parameters that follow define rainflow cycle counting parameters for rainflow data reduction. See Remark 5.   |
| RTYPE      | Method of rainflow data reduction (Time History Compression). Value can be "LOAD" for load time history data reduction on each load time history or "FAST" for performing a less accurate, but computationally faster method (Character; no default). See footnote in Table 9-17.   |
| GATE       | Load value used as gate range. This load is used for gating out small disturbances of "noise" in the time history using a peak-valley extraction method to speed up the analysis. Only used if RTYPE=LOAD. GATE and PCTRED are mutually exclusive. PCTRED ignored if GATE is supplied. (Real $\geq 0.0$ , Default=0.0 )   |
| PCTRED     | Percent reduction value based on the maximum load range used to reduce the load time history using a peak-valley extraction method to speed up the analysis. Only used if RTYPE=LOAD. Ignored if GATE is supplied. GATE and PCTRED are mutually exclusive. (0.0 $\leq$ Real $\leq 100.0$ , Default=50)  |
| "CERTNTY"  | <b>SOL 101, 103, 112</b> only. Flag indicating that the parameter that follows defines the certainty of survival in fatigue analysis.   |
| SURV       | Certainty of survival based on the scatter in the S-N or e-N curves. (0.1 $\leq$ Real $\leq 99.9$ ; Default = 50). See Remark 7.  |
| "FOS"      | <b>SOL 101, 103, 112</b> : Flag indicating that parameters that follow are used in a factor of safety analysis. The presence of this flag triggers a factor of safety analysis. See Remark 8. and 14.   |
|            | <b>SOL 108, 111</b> : Flag indicating that the total signal time duration desired is to be specified for use in calculating margin of safety. Only the LIFE field is required in this case and defaults to 1.0. A margin of safety is always calculated.  |
| OPTION     | Supported option is LIFE, requesting a life-based factor of safety analysis or NONE. (Character; default = LIFE).   |

| Descriptor | Meaning  |
|------------|--|
| LIFE       | <b>SOL 101, 103, 112:</b> The targeted design life given in user defined life units (such as laps, miles, etc.) as defined by “UNITS” line on FTGSEQ or FTGLOAD entry, or <i>Repeats</i> of the cyclic loading if no “UNITS” line exists (Real>0, no default).<br><br><b>SOL 108, 111:</b> Total time duration required for margin of safety calculation in time units specified by TUNIT field on FTGSEQ entry. |
| BACKACC    | <b>SOL 101, 103, 112 only.</b> The back calculation accuracy used to control back calculation iterations that determine the scale factor on the applied stress level to achieve the target design life. Defined as a percentage error on the target design life. (0.01 < Real ≤100.0, Default = 1.0).  |
| MAXFAC     | <b>SOL 101, 103, 112 only.</b> The maximum safety factor to calculate and report. When this threshold is exceeded, the analysis will go on to the next element and report the maximum for the exceeded element (2.0 ≤ Real ≤ 5.e6, Default = 5.0).   |
| MINFAC     | <b>SOL 101, 103, 112 only.</b> The minimum safety factor to calculate and report. If the result is below this threshold, the analysis will report MINFAC as the safety factor for this element and go on to process the next element. (0.0<Real≤0.5, Default = 0.2).   |
| "DAMAGE"   | <b>SOL 101, 103, 112 only.</b> Flag indicating that parameters for static failure check are to follow.   |

| Descriptor | Meaning  |
|------------|--|
| CHECK      | Action to take on static failure. 0, 1, or -1 for Warn, Stop, or None, respectively. Default = 0. See Remark <a href="#">9</a> . <ul style="list-style-type: none"> <li>■ 0=Warn: the calculation continues but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the FAILDAM field, 2.0, by default. Not recommended for fatigue analysis of spot welds.</li> <li>■ 1=Stop: The analysis stops the damage calculation for an entity with static failure and continues on to the next entity. At the end of the entire analysis of all entities, if there has been a static failure, a fatal message is issued and the job is terminated. This setting is not recommended for fatigue analysis of spot welds. Fatigue output ( CSV, FER, etc ) that was requested is still kept, and entities that failed the UTS test have damage/life set to 2/0.5 (assuming the FTGPARM/FAILDAM is set to default value).</li> <li>■ -1=False: No static failure check is made. Damage will be calculated up to the numerical limit defined by the MAXDAM field. Note that this may be using an extrapolation of the S-N curve and may predict damage values per cycle greater than 1.0. This also has an additional effect using standard SN parameters. In this case, the adjustment of the S-N curve below 1000 cycles to take into account the static strength of the material is disabled.</li> </ul> |
| FLOOR      | This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant as similar functionality may be accessed using the fatigue cutoff Nfc in the material definition. ( $0.0 \leq \text{Real} \leq \text{MAXDAM}$ ; Default = 0.0).  |
| MAXDAM     | This parameter sets a numerical upper limit on the damage that can be predicted for each cycle. Setting this to >1 shows how much unacceptable damage has accumulated. ( $\text{FLOOR} \leq \text{Real} \leq \text{FAILDAM}$ ; Default = 1.0).   |
| FAILDAM    | This parameter assigns the numerical value of damage for any entity (node or element) when the stress exceeds the UTS. This parameter is used to identify static failure from a single cycle when looking at damage. ( $1.0 \leq \text{Real} \leq 1e30$ and $\text{MAXDAM} \leq \text{Real}$ ; Default = 2.0).   |
| "SPOTW"    | SOL 101, 103, 112 only. Flag indicating that parameters for fatigue analysis of spot welds are to follow. See Remark <a href="#">10</a> .  |
| COMB       | Stress combination to use in the fatigue analysis. Acceptable values are listed in <a href="#">Table 9-17</a> after the Remarks below (Character; Default=STNDRD, which is basically a critical plane analysis).   |

| Descriptor | Meaning   |
|------------|---|
| CORR       | Mean stress correction to use in the fatigue analysis of spot welds. Only NONE or SIMPLE are valid for fatigue analysis of spot welds. See <a href="#">Table 9-18</a> and <a href="#">Table 9-19</a> below. (Character; Default = NONE).  |
| NANGLE     | The number of calculation angles in 360 degrees around the spot weld. (10≤Integer≤360; Default = 18, i.e., every 20 degrees)  |
| SWLOC      | Location on the spot welds to report fatigue life. Zero (0) reports worst case angle and location (top/bottom sheet or nugget); one (1) reports worst case angle for each location; two (2) reports worst case location for each angle; three (3) reports all locations and angles. (Integer 0, 1, 2, or 3, Default=0).           |
| MIDDLE     | Whether to process middle sheets if there are more than two sheets in the weld. (Integer 0 or 1, Default = 0 - do not process middle sheets). See Remark <a href="#">12</a> .   |
| TORSION    | Whether to calculate torsion in the spot weld. (Integer 0 or 1, Default = 0 - do not calculate torsion). See Remark <a href="#">13</a> .  |
| "SEAMW"    | <b>SOL 101, 103, 112</b> only. Flag indicating that parameters for fatigue analysis of seam welds are to follow. See Remark <a href="#">10</a> .  |
| COMB       | Stress/strain combination to use in the fatigue analysis of seam welds. Only ABSMAXPR and CRITICAL are supported. See <a href="#">Table 9-17</a> below. (Character; Default = ABSMAXPR).  |
| CORR       | Mean stress correction to use in the fatigue analysis of seam welds. Only NONE or FKM are valid for fatigue analysis of seam welds. See <a href="#">Table 9-18</a> and <a href="#">Table 9-19</a> below. (Character; Default=NONE).   |
| THICK      | Thickness correction to be applied. (Integer 0 or 1; Default =0 - no correction)  |
| LOCSM      | Location on seam weld to report life. Valid values are "NODE", "SGAGE", "CORNER", "BILIN", or "CUBIC" (Character; Default = NODE). If "STRESS" line is also included, LOCSM must be the same as LOC. LOC=ELEM is not valid for fatigue analysis of seam welds and cannot be mixed with LOC = NODE. See Remark <a href="#">4</a> . |
| RESENT     | Result entity type used in the fatigue analysis of seam welds; only STRESS is currently supported. (Character; Default = STRESS).   |
| "MULTI"    | <b>SOL 101, 103, 112</b> only. Flag indicating that parameters for biaxial/multiaxial assessment are to follow. See Remark <a href="#">11</a> . and <a href="#">14</a> .  |

| Descriptor                    | Meaning  |
|-------------------------------|--|
| MMTHD                         | Can be set to the following: <ul style="list-style-type: none"> <li>■ NONE = No multiaxial assessment is done.</li> <li>■ SIMPLE = Calculates simple biaxiality ratios only.</li> <li>■ STANDard =Standard method of assessment, which merely returns the results of the assessment.</li> <li>■ AUTO = Performs the standard method, but then may recalculate fatigue damage depending on the results of the assessment.</li> </ul>  |
| NONLWR<br>NONUPR              | Used only when MMTHD = AUTO. Lower and Upper thresholds used to check if the loading is proportional. This is used in combination with the biaxiality ratio thresholds.<br>NONLWR: (0<Real≤NONUPR, Default = 0.25)<br>NONUPR: (NONLWR<Real, Default = 0.5)<br>NONUPR not used if TYPE=SN   |
| BIAXLWR<br>BIAXMID<br>BIAXUPR | Used only when MMTHD = AUTO. Lower, Middle, and Upper threshold ratios used to check if the loading is proportional. This is used in combination with the non-proportionality factor thresholds.<br><br>For TYPE=EN:<br>BIAXLWR: (-1≤Real≤BIAXMID, Default = -0.6)<br>BIAXMID: (BIAXLWR≤Real≤BIAXUPR, Default = 0.25)<br>BIAXUPR: (BIAXMID≤Real≤1, Default = 0.6)<br><br>For TYPE=SN:<br>BIAXLWR: (-1≤Real≤0, Default = -0.6)<br>BIAXMID: Not used<br>BIAXUPR: (0<Real≤1, Default = 0.6) |
| ZERO                          | Used only when MMTHD = AUTO. Stress range below which damage is assumed zero and therefore no recalculation is performed. Specified as a percentage of UTS. (0<Real≤100, Default=10.0).  |
| GATE                          | Used only when MMTHD = AUTO. This gate value is used to prevent small stresses from adversely affecting the biaxiality calculation. Stresses below this value are not included in the biaxiality calculation; their biaxiality ratio will be set to zero (0). The value is set as a percentage of UTS. (0<Real≤100, Default=20.0).   |

| Descriptor | Meaning  |
|------------|--|
| "NAVG"     | <b>SOL 101, 103, 112</b> only. Flag indicating that parameters for nodal (grid point) stress (or strain) averaging are to follow. The LOC field must be set to NODA, otherwise this line is ignored. These parameters are the same as those specified in the SURFACE and VOLUME case control for requesting grid point stresses/strains and the user is referred to the OUTPUT(POST) section for more details. All defaults are used if this line is omitted. See Remark 15.   |
| MTHD       | Method to calculate average grid point stresses/strains: <ul style="list-style-type: none"> <li>■ TOPO = Topological method (default)</li> <li>■ GEOM = Geometric method.</li> </ul>   |
| OUTPUT     | Coordinate system in which to translate stresses/strains before averaging: <ul style="list-style-type: none"> <li>■ -1 : Specifies the element coordinate system for output</li> <li>■ 0 : Specifies the basic coordinate system for output (default)</li> <li>■ CID : Specifies the coordinate system defined on a CORDij bulk data entry for output</li> </ul> <p>The X-axis of this system is used to define the X stress direction. For shell elements the X-axis of the specified coordinate system is projected onto the surface or, that is, the resulting averaged stresses are a projection onto the surface.</p> |
| NORMAL     | For shell elements defining a surface, this specifies the method to define the surface normal. For solid elements defining a volume, this specifies the method to define the stress Z-axis. Indirectly this also sets the reference direction for positive fiber and shear stress output, but has no effect when OUTPUT=-1. Can be set to R (default), X1, X2, or X3.  |
|            | M Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter. R Specifies the radius vector from the origin of reference coordinate system to the grid point.   |
|            | The Y stress direction is the cross product of the Z and X stress directions defined by the NORMAL and the X-axis of the OUTPUT coordinate system.   |
| "VIBFTG"   | <b>SOL 108, 111</b> only. Flag indicating that vibration fatigue parameters follow. The line is optional and defaults are used if not present.   |
| ATYPE      | Analysis method to use to generate the PDF of rainflow ranges: DIRLIK, NARROW, STEIN, SINES. (Character; default = DIRLIK)   |

| Descriptor | Meaning  |
|------------|--|
| MAXSTR     | Maximum stress to use as a function of the number of RMS stress levels to determine maximum stress in the rainflow cycle count. (Real>0.0; Default=10.0).  |
| CLIPLVL    | Value of stress where all stresses are "clipped" (kept at that level) when doing fatigue life calculation. Real>0.0; (Default CLIPLVL=MAXSTR). CLIPLVL=6 means +/- 3*RMS amplitudes or 6*RMS in terms of range.  |
| MAXPEAK    | Used to calculate an estimation of maximum and minimum stress (number of RMS levels) in the random response. It is not used in any way to calculate damage or life. This is also used when the maximum elastic-plastic peak strain is calculated. Where a mean load is included the peak stress is calculated as the mean_stress + MAXPEAK*RMS and the minimum stress is calculated as the mean_stress - MAXPEAK*RMS. This is similar to the time domain approach where the maximum and minimum stress are retained from the rainflow cycle count. (Real>0; Default=CLIPLVL/2 or, if CLIPLVL not defined, MAXSTR/2). |
| STRBINS    | Number of bins to generate in rainflow cycle count. Default = 1280 for ATYPE=SINES otherwise 32. Not recommended to use anything over 5000.  |
| MAXFREQ    | Percentage of frequency content retained as a function of the first spectral moment $m_0$ of the PSD defining the maximum frequency used to integrate the spectral moments for $m_1$ , $m_2$ , and $m_4$ . 0<=Real<=100.0; Default=99.9. 100% is not a recommended setting as it can cause numerical instability related to the calculation of the 4th spectral moment.  |
| NCALC      | Alternative options for the way "N" (life to failure) is calculated for a given stress bin. Valid values are MID, UPR, or AVG for middle, upper or average. (Character; Default=AVG). See <a href="#">Table 9-20</a> .   |

**Remarks:**

1. FTGPARM bulk data will be ignored if not selected by a FATIGUE case control entry. If a FTGPARM entry is not defined, default properties are used for the requested fatigue analysis.
2. **SOL 101, 103, 112:** For total life or stress-life (TYPE=SN), only STRESS results can be used. For crack initiation or strain-life (TYPE=EN), the fatigue analyzer may use either STRESS or STRAIN results from the finite element analysis. This selection should make no difference to the final results of a crack initiation calculation, as strains will always be calculated. The exception is when shell results are used. In this case, STRESS should be selected because only 2D results are available and the absence of the out-of-plane strains will cause incorrect calculation of combined parameters. It is an error to have both STRESS and STRAIN lines. If both are missing, then STRESS will be assumed with its default values.
3. **SOL 101, 103, 112:** PLAST can be set to NEUBER or SEEGER. Please note that NEUBER can be used universally for uniaxial stress states. SEEGER requires a 2D stress state and is generally used when the stress state is not purely uniaxial. PLAST is only valid for TYPE=EN.

4. If LOC=ELEM (element center), fatigue lives are calculated based on stresses/strains at element centroids (not recommended for anything but shell or 2D solid elements (plane stress/strain and axisymmetric) and the RECOVER field is ignored or automatically set to CENTER.

If LOC=NODE, the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. Note that for LOC=NODE, the RECOVER field can be set to SGAGE, BILIN, CUBIC, or CORNER to control which stresses are used in the fatigue analysis for element nodal stresses and correspond to the same options as on the STRESS case control.

If LOC is set to NODA, the same RECOVER options as LOC=NODE apply, however:

**SOL 101, 103, 112:** With LOC=NODA, fatigue lives are based on stresses/strains from nodal averaged (grid point) stresses, resulting in a single damage value at each node as opposed to using LOC=NODE where the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. The grid point stresses are generated using Nastran's GPSTRESS output request internally (it is not required to include the GPSTRESS case control unless you wish to view the grid point stress also). See the NAVG line settings also on how to control grid point stress calculations.

**SOL 108, 111:** With LOC=NODA, after the nodal transfer function (TF) stresses for each element contribution are converted to the requested stress combination (e.g. COMB=SGVON), the stresses are summed and averaged at each node. This is done at every frequency to give a single TF of stress at each node. Thus a single damage and fatigue life per node is reported as opposed to slightly different values when LOC=NODE. Please note that the coordinate system in which the elements are defined should be consistent (the same). Otherwise the nodal averaging will not be done using consistent coordinate systems. Thus, for solid element, setting CORDM on the PSOLID entry to the Basic coordinate system or a specified CID is required and setting CID=-1 or -2 (element systems) is not recommended. For shell elements, MCID should be set on the respective element entry.

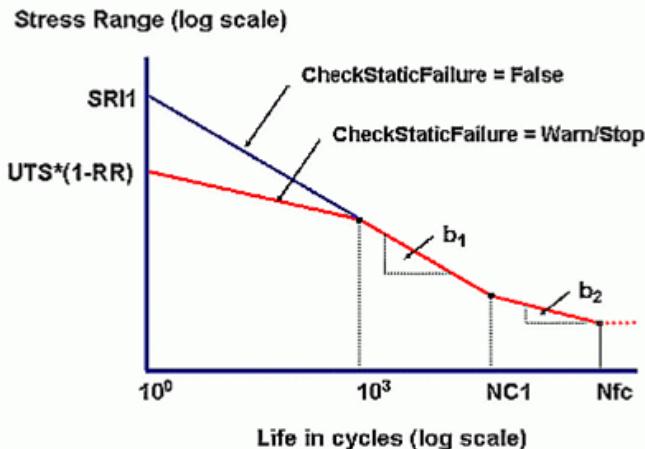
This parameter cannot be changed on a RESTART since it would require a different set of stress/strain data, which is not allowed on a RESTART for a pure fatigue analysis.

LOC=NODE generally gives more conservative answers than LOC=NODA since no averaging of the stresses/strains is done from the contributions of the surrounding elements.

5. **SOL 101, 103, 112:** SRESOLVE is applicable for all three values of LOC, i.e.: LOC = ELEM, NODE or NODA. SRESOLVE is an option to evaluate surface stresses instead of volume stresses. When surface resolved stresses are requested from 3D solid elements, a thin layer of 2D shell elements is internally created encompassing the volume of the elements defined on the FTGDEF entry from which the 2D stress state is then determined. This parameter is necessary if a multiaxial assessment of a model made of 3D elements is necessary, otherwise 3D elements are ignored during a multiaxial assessment. Fatigue life and multiaxial statistics are only reported for these newly defined 2D elements (or reduced set of nodes) on the surface of the volume (free surface). Interior nodes are not included in the fatigue calculation. The internal element IDs generated by this option can be controlled by using MSC Nastran SYSTEM cell 183 (Default is set to 200000000).

Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses. Otherwise a fatal condition is flagged and the analysis stops.

6. This RAINFLOW flag is only necessary to use if it desired to speed up the analysis for purposes of quick critical location identification or for sanity checks. RTYPE determines how to compress the input time history data for the purposes of speeding up the calculation, possibly at the loss of accuracy of the results. RTYPE=FAST simply reduces the load time histories to a single cycle using only the minimum and maximum from the input time series data, without losing phase information across multiple load cases. RTYPE=LOAD using PCTRED performs a multi-channel peak-valley extraction on the time series data prior to processing with PCTRED value set as a percentage of the range of the input data. RTYPE=LOAD using GATE performs a multi-channel peak-valley extraction on the time series data prior to processing with GATE value applied directly in the units of the time series data.
7. Certainty of survival is based on scatter in the S-N or  $\varepsilon$ -N curves. It is used to modify the curves according to the standard error parameters ( $SE_n$ ) defined in the MATFTG material entry. A higher reliability level requires a larger certainty of survival.
8. SOL 101, 103, 112: This FOS option will calculate a type of safety factor for over design analysis to be performed. This analysis is in addition to the normal fatigue life/damage output and must be requested by the presence of this FOS flag and its parameters. This analysis method can be very useful for those components which predict infinite life, providing a measure of the risk of fatigue failure. The results of this analysis are factors by which the stress would have to be scaled to attain the specified design life. A value of one suggests that the specified life will be exactly attained whereas a factor less than one means the desired life will not be attained. Factors greater than one are, therefore, most desirable. By definition the resulting life values will be the target life, thus only the scale factor and maximum/minimum stress results are of interest when FOS is defined.  
**SOL 108, 111:** This FOS option is actually a margin of safety (MOS) calculation, which requires only the desired or required duration of entire loading sequence in time units. By default this is set to 1.0 and the time units are those defined by the TUNIT field on the FTGSEQ entry.
9. The static failure check has not had any validation in regards to spot welds and any prediction of static failure should be treated with caution. Also, setting CHECK=0 or 1 (Warn or Stop) will result in the S-N curve being adjusted in the low cycle region (< 1000 cycles) in the same way as for the standard S-N method, as illustrated in the figure below. This is not desirable for spot weld fatigue. It can be avoided, either by setting CHECK = -1 (recommended), or setting a value of the UTS such that  $UTS*(1 - RR) = SRI1$  or greater.



10. For fatigue analysis of spot and seam welds, besides the “SPOTW” and “SEAMW” lines themselves, “RAINFLOW”, “CERTNTY”, and “FOS” are also applicable. “MULTI” is applicable to seam weld, but not spot welds. All others are ignored. TYPE field is ignored as an SN analysis is forced for fatigue of spot and seam welds.
11. Biaxial/Multiaxial assessment can be requested for all but fatigue analysis of spot welds and is ignored by SOL 200 optimization runs. These assessments require a 2D state of stress. For this reason only shell elements are supported with this feature. MMTHD=AUTO is not allowed for fatigue analysis of seam welds.
12. The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. For this reason, MIDDLE is set off by default to omit any fatigue calculation on the middle sheets, which might otherwise give a false positive failure prediction.
13. Setting the TORSION option enables fatigue calculations to be made based on the shear stress around the periphery of the spot weld in each sheet due to the torsional load on the spot weld. Please note the following:
  - a. This option requires that suitable S-N curves be defined for the torsion case, which may mean a separate run for torsion vs for a non-torsion run.
  - b. Do not use this option on unless your modeling strategy can generate realistic torsional loads in the spot welds. For example, this is not true of the simple CBAR modeling approach.
  - c. The mean stress correction is applied a little differently, because the sign of a mean shear stress cannot influence the fatigue damage. For this reason, the ABSOLUTE value (i.e., the magnitude) of the mean stress is used when applying any mean stress correction for the torsion case.
  - d. The calculation is a completely separate operation—there is no combination of the torsional stresses with the other stresses on the spot weld.

- e. This option has had little validation, and no suitable S-N curves are provided with the software. However, it has been successfully used in practice, using S-N curves deduced from practical experience with car body structures, and has proved a useful tool for predicting premature failure in the small number of spot welds that may experience significant torsional loads.
  - f. The results of the torsional fatigue calculation are identified in the results by being associated with an angle of -1.
14. Factor of Safety (FOS) analysis is only compatible with the MMTHD=SIMPLE if doing a multi-axial/biaxial (MULTI) assessment in the same run. MMTHD=AUTO is not allowed in fatigue analysis of seam welds (SEAMW). The stress/strain combination (COMB) is ignored if MMTHD=AUTO because the program will determine whether to use a critical plane or absolute maximum principal combination method.
15. When LOC=NODA for requesting nodal average stresses, the grid point stresses generator is used to determine the nodal averaging. A limited set of controls is provided on the NAVG line to control how the averaging method. Internally, SURFACE and/or VOLUME definitions are created for the grid point stress request, just as if the GPSTRESS case control, with its associated OUTPUT(POST) commands, were given. However, only one SURFACE and/or VOLUME is determined based on the elements defined on the FTGDEF entry for requesting fatigue analysis. The user should be aware of possible stress discontinuities that can result from this.
16. By default the response PSD at the most critical location is output using LOGLVL=0 via a <jobname>PSD.csv file. If LOGLVL=1, response PSDs for all requested locations are output to that same file. Also a <jobname>RCC.csv file is produced with limited data showing the summation of damages from the rainflow cycle count. If LOGLVL=2, both files are created and in the latter, addition the stress and damage BIN data for each event is included. Be aware that this can create a lot of data, resulting in huge output files, and can severely degrade performance. Please use this option with care and limit the output to only the critical elements of interest.

Table 9-17 Allowable Values for the COMB Field

| Stress/Strain Combination                             | SOL 101, 103, 112* Valid for TYPE | SOL 108, 111 Valid for TYPE | Meaning  |
|---|-----------------------------------|-----------------------------|--|
| ABSMAXPR  | SN (+SeamWeld)<br>EN              | SN<br>EN                    | Absolute Maximum Principal (default)†  |
| MAXPRINC  | SN                                | SN<br>EN                    | Maximum Principal Stress‡  |
| SGVON   | SN<br>EN                          | SN<br>EN                    | Signed von Mises Stress (recommended for SOL 108, 111 but not default)   |
| VONMIS  | SN                                |                             | von Mises Stress   |
| SGMAXSHR  | SN<br>EN                          |                             | Signed Maximum Shear   |
| MAXSHR  | SN                                |                             | Maximum Shear  |
| CRITICAL  | SN (+Seam Weld),<br>EN            |                             | Critical Plane Analysis - every 10 degrees by default unless NANGLE is specified otherwise for standard SN/EN analysis only- Not valid for 3D solid or CSHEAR elements - element must have an in-plane stress state. |
| STNDRD  | SN<br>(Spot Weld only)            |                             | This is basically a critical plane analysis with calculations being done at NANGLEs around the circumference of the spot weld.   |
| COMPX<br>COMPY<br>COMPZ<br>COMPXY<br>COMPYZ<br>COMPZX |                                   | SN<br>EN                    | Individual stress components   |

\* In time domain, six multiaxial components of the stress tensor are resolved into one uniaxial, combined value for fatigue calculations at each entity per time step since damage models are based on uniaxial theories. For S-N analysis, the signed von Mises (SGVON) will be smaller than the Absolute Maximum Principal (ABS MAXPR) when there is positive biaxiality and hence this selection would be less conservative. (Note also that some BS weld classes require shear stress to be used.) The signed parameters uses the sign of the absolute maximum principal value for conservative fatigue life estimates. It is not recommended to use non-signed values (MAXPRINC, VONMIS, MAXSHR). If using these, do not set the RTYPE=FAST on the 'RAINFLOW' line as this results in no cyclic stress cycling.

† In frequency domain, this is a Maximum Principal calculation using accurate phase scanning.

‡ In frequency domain, this is a fast Maximum Principal calculation using an approximate algorithm and is less accurate than ABSMAXPR, but improves computational speed.

Table 9-18 Allowable Values for the CORR Field

| Mean Stress Correction | SOL 101, 103, 112<br>Valid for TYPE | SOL 108, 111<br>Valid for TYPE | Meaning  |
|------------------------|-------------------------------------|--------------------------------|--|
| NONE                   | SN (+Spot/Seam Weld)<br>EN          | SN<br>EN                       | No mean stress correction (default)  |
| GOODMAN                | SN                                  | SN<br>EN                       | Goodman mean stress correction   |
| GERBER                 | SN                                  | SN<br>EN                       | Gerber mean stress correction  |
| GDMANT                 | SN                                  | SN<br>EN                       | Tension only Goodman mean stress correction  |
| GRBERT                 | SN                                  | SN<br>EN                       | Tension only Gerber mean stress correction   |
| FKM                    | SN (+Spot/Seam Weld)                |                                | FKM mean stress correction method.<br>Uses M1, M2, M3, M4 slopes as defined in <a href="#">MATFTG</a> entry. See remarks in <a href="#">MATFTG</a> entry.  |
| SIMPLE                 | SN (Spot Weld only)                 |                                | Modified FKM mean stress correction method where M1=M2=M3=M4=-MSS as defined in the <a href="#">MATFTG</a> entry.  |
| INTERP                 | SN                                  |                                | Interpolation method used with multiple SN curves only. TYPE field of <a href="#">MATFTG</a> entry must be set to MEAN, RRATIO, or LIFE. Requires that there be multiple curves defined, one at R=-1 for TYPE=RRATIO, or zero (0) mean stress for TYPE=MEAN. |
| SWT                    | EN                                  | SN<br>EN                       | Smith-Watson-Topper mean stress correction   |
| MORROW                 | EN                                  | EN                             | Morrow mean stress correction  |
| WALKER                 |                                     | EN                             | Walker mean stress correction  |
| MMPDS                  |                                     | SN<br>EN                       | Walker mean stress correction built into the MMPDS-01 material curves  |

Table 9-19 Allowable S-N vs. Mean Stress Correction Methods for SOLs 101, 103, 112

| Mean Stress Correction | S-N Method |                  |                     |       |            |
|------------------------|------------|------------------|---------------------|-------|------------|
|                        | Standard   | Multi Mean Curve | Multi R-Ratio Curve | Haigh | Bastenaire |
| NONE                   | YES        | YES*             | YES†                | NO    | YES        |
| GOODMAN                | YES        | YES*             | YES†                | NO    | YES        |
| GERBER                 | YES        | YES*             | YES†                | NO    | YES        |
| GDMANT                 | YES        | YES*             | YES†                | NO    | YES        |
| GRBERT                 | YES        | YES*             | YES†                | NO    | YES        |
| FKM                    | YES        | YES*             | YES†                | NO    | YES        |
| SIMPLE‡                | YES        | NO               | NO                  | NO    | NO         |
| INTERP                 | NO         | YES              | YES                 | YES   | NO         |

\* Allowed but a curve at zero mean stress must be present.

†Allowed but a curve at  $R = -1$  must be present.

‡Spot Welds only.

Table 9-20 Alternative NCALC options to calculate “N” (cycles to failure)

| NCALC option | Meaning   |
|--------------|---|
| AVG          | Calculation of “N” is done by adding up all the values of N within the BIN and then dividing through by the number of N values used. In this context the average value of N is calculated over the BIN. |
| MID          | “N” is extracted at the mid (stress) point of the BIN.  |
| UPPER        | “N” is extracted at the upper (stress) point of the BIN.  |

## FTGSEQ

## Fatigue Load Sequence

Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or vibration fatigue using SOL 108 or SOL 111.

Format (SOL 101, 103, 112):

| 1      | 2       | 3       | 4      | 5   | 6    | 7   | 8    | 9   | 10 |
|--------|---------|---------|--------|-----|------|-----|------|-----|----|
| FTGSEQ | ID      | EVNTOUT | METHOD |     |      |     |      |     |    |
|        | FID1    | N1      | FID2   | N2  | FID3 | N3  | FID4 | N4  |    |
|        | FID5    | N5      | ...    | ... | ...  | ... | ...  | ... |    |
|        | -etc.-  |         |        |     |      |     |      |     |    |
|        | "UNITS" | EQUIV   | EQNAME |     |      |     |      |     |    |

Format (SOL 108, 111):

| 1      | 2       | 3       | 4      | 5     | 6    | 7   | 8    | 9   | 10 |
|--------|---------|---------|--------|-------|------|-----|------|-----|----|
| FTGSEQ | ID      | EVNTOUT |        | TUNIT | LDM  |     |      |     |    |
|        | FID1    | N1      | FID2   | N2    | FID3 | N3  | FID4 | N4  |    |
|        | FID5    | N5      | ...    | ...   | ...  | ... | ...  | ... |    |
|        | -etc.-  |         |        |       |      |     |      |     |    |
|        | "UNITS" | EQUIV   | EQNAME |       |      |     |      |     |    |

Examples:

|        |   |    |  |  |  |  |  |  |  |
|--------|---|----|--|--|--|--|--|--|--|
| FTGSEQ | 1 |    |  |  |  |  |  |  |  |
|        | 6 | 15 |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique ID with respect to all other FTGSEQ and FTGEVNT entries called out by a FATIGUE case control. (Integer>0) See Remark 1.   |
| EVNTOUT    | Flag for requesting fatigue output for each event individually. (Integer; 0=no or 1=yes; Default = 0). See Remark 7. For SOL 108 and 111, this also controls the written output request for spectral statistics (m0, m1, m2, m4, etc.).                                    |
| METHOD     | SOL 101, 103, 112 only. Event processing method. 0=independent, 1=Combined Full, 2= Combined Fast (Integer; Default = 0). This entry is only honored from the controlling FTGSEQ entry when called out by the FATIGUE case control. Otherwise it is ignored. See Remark 2. |
| TUNIT      | SOL 108, 111 only. Time units of Ni values. SECS, MINS, HRS, DAYS, WKS, MTHS, YRS for seconds, minutes, hours, days, weeks, months, or years, respectively (Character; Default=SECS).  |

| Descriptor | Meaning  |
|------------|--|
| LDM        | <b>SOL 108, 111</b> only. FE load magnitude used to create transfer function. Used to normalize the transfer function stresses (Real, Default=1.0).  |
| FIDi       | <ul style="list-style-type: none"> <li>■ <b>SOLs 101, 103:</b> ID of a FTGEVNT or another FTGSEQ entry for pseudo-static fatigue analysis using SOL 101 or a modal analysis using SOL 103 (Integer &gt; 0). See Remark 3. and 4.</li> <li>■ <b>SOL 112:</b> A subcase ID that represents the loading event or another FTGSEQ entry for modal transient fatigue analysis using SOL 112. (Integer &gt; 0). See Remark 3. and 4.</li> <li>■ <b>SOL 108, 111:</b> ID of a FTGEVNT entry for random vibration fatigue using SOLs 108 or 111. Nested FTQSEQs are not allowed.</li> </ul>   |
| Ni         | <ul style="list-style-type: none"> <li>■ <b>SOLs 101, 103, 112:</b> Number of repeats of this loading sequence or event (Real&gt;0.0, Default=1.0). Ignored if only one event is defined. For METHOD equal 1 and 2, Ni must be a whole number, i.e., 1.0, 2.0, 3.0, etc. In other words, fractions of events are not allowed. Fractional events are also not allowed for any METHOD if the corresponding FID references a FTGSEQ entry. See Remark 3. and 4.</li> <li>■ <b>SOL 108, 111:</b> Duration of each event in TUNIT units. This value is not used for random loading only. These values are also not needed and overwritten if SWRATE/NSWEEP are provided on a FTGLOAD entry of TYPE= SINE (sine sweep). (Real&gt;0.0; Default=1.0).</li> </ul> |
| "UNITS"    | Flag indicating that a fatigue equivalent unit name is applied to this loading. See Remark 5. and 6.   |
| EQUIV      | Number of equivalent units. (Real>0.0; Default=1.0 ). See Remark 5. and 6.   |
| EQNAME     | Equivalent name of this loading event. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remark 5. and 6.   |

### Remarks:

1. FTGSEQ bulk data are called out by FATIGUE case control.
2. Processing of events can be done by determining the damage due to each event independently (default) and then summing the damage due to all events. Or the events can be concatenated and damage determined after rainflow cycle counting over all events. The advantage of the independent method over the combined methods is computational expense versus accuracy. The combined method will close all cycles, whereas the individual method may miss a large damaging cycle if the cycle begins in one event and ends or closes in a subsequent event. The combined fast method performs a rainflow count data reduction to speed up the analysis and determine the most critical locations first and then redoes a full analysis on the critical locations.

3. Once a FTGSEQ bulk data entry is referenced in an FIDi field, it can't be referenced again in any other FTGSEQ entries (within its own associations - the same fatigue analysis) to avoid infinite loops. And if it is referenced by the FATIGUE case control, it cannot appear in any FIDI field of the FTGSEQ bulk data.
4. Different FTGEVNTs can be set up and the user can construct each sequence by specifying how many times to repeat each event in a sequence. One sequence could then be referenced in another sequence to tell the new sequence how many times to repeat that sequence. As an example, assume there are three events an automobile is subjected to: cobble stones, pot holes, speed bumps. One sequence might be five (5) "cobble stones," six (6) "potholes" and three (3) "speed bumps." This sequence may be called "torture track." Also define two more events called "cornering left" and "cornering right." A load sequence of ten (10) "cornering left" and ten (10) "cornering right" might be called "country road." Now with a nested FTGSEQ you can put these together any way you want. So one fatigue analysis might use a sequence of only "country road," another of only "torture track" and another of a combined six (6) "torture tracks," five (5) "country roads," followed by one (1) more "torture track" and one (1) more "country road" This would result in 3 fatigue analyses as shown in [Table 9-21](#) below.
5. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the resulting stress time history sequence. Equivalent units specified on FTGLOAD entries are ignored when FTGSEQ entries are used
6. Example of using equivalent units: If one repeat (or total duration) of the sequence is equivalent to five (5) times around a test track, the equivalent unit name, EQNAM, might be "laps," and the equivalent unit, EQUIV, would be five (5). Fatigue life will be reported in these units if defined, otherwise they are reported as repeats of the loading sequence. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
7. **SOL 101, 103, 112:** For duty cycle jobs, temporary time series channel files are created. Add 10 to the EVNTOUT value if you wish to keep the files (.dac or .s3t extension), otherwise they are deleted after the job terminates. For SOL 200 only EVNTOUT=0 is supported. Also some additional output file formats are not supported if EVNTOUT>0. See the [FATIGUE case control entry](#) for STROUT=4 limitations.
8. **SOL 108, 111:** For vibration fatigue a FTGSEQ cannot reference both an event with single input random loading and multiple input random loading.

**Table 9-21      Example of nested FTGSEQ entries**

```

SET 1=22,33,44
FATIGUE=1
BEGIN BULK
$
FTGSEQ,22      $<-- 1 country road only
,2
FTGSEQ,33      $<-- 1 torture track only
,3
$combined
FTGSEQ,44      $<-- 6 torture track, 5 country road, + 1 of each
,3,6.0,2,5.0,3,1.0,2,1.0
$country road:
FTGSEQ,2
,8,10.0,9,10.0   $<-- 10 cornering left + 10 cornering right

```

```
$torture track:  
FTGSEQ 3  
    ,5,5.0,6,6.0,7,3.0 $<-- 5 cobble stones, 6 potholes, 3 speed bumps  
$  
FTGEVNT,5...    $cobble stones  
FTGEVNT,6...    $potholes  
FTGEVNT,7...    $speed bumps  
  
FTGEVNT,8...    $cornering right  
FTGEVNT,9...    $cornering left
```

**GBAG****Gas-Bag Pressure Definition**

Defines the pressure within an enclosed volume. Used in SOL 700 only.

**Format:**

| 1    | 2     | 3      | 4       | 5        | 6     | 7       | 8     | 9     | 10 |
|------|-------|--------|---------|----------|-------|---------|-------|-------|----|
| GBAG | GID   | BSID   | TRIGGER | TRIGGERV | PORID | INFID   | HTRID | INTID |    |
|      |       |        |         |          |       |         |       |       |    |
|      |       |        |         |          |       |         |       |       |    |
|      | CPGAS | PGPASV | RGAS    | PENV     |       | REVERSE | CHECK | PINIT |    |
|      | TINIT | TENV   |         |          |       |         |       |       |    |

**Examples:**

|      |          |       |       |         |    |    |    |  |  |
|------|----------|-------|-------|---------|----|----|----|--|--|
| GBAG | 101      | 37    |       |         | 11 | 12 |    |  |  |
|      |          |       |       |         |    |    |    |  |  |
|      |          |       |       |         |    |    |    |  |  |
|      | CONSTANT | 1000. | 297.0 | 101325. |    | ON | ON |  |  |
|      | 293.0    |       |       |         |    |    |    |  |  |

| Descriptor | Meaning  |             |                            |  |
|------------|--|-------------|----------------------------|--|
| GID        | Unique gas-bag number.   | I > 0       | Required                   |  |
| BSID       | Identification number of a BSURF, BCBOX, BCPROP, BCMATL, or BCSEG entry defining the surface of the gas-bag.                       | I > 0       | Required                   |  |
| TRIGGER    | The time-dependent parameters are offset in time.<br>TIME The offset is defined at TRIGGERV.                                       | Character   | TIME                       |  |
| TRIGGERV   | The value of the offset in time.   | Real        | Required                   |  |
| PORID      | Number of a set of LEAKAGE entries, that defines the porosity (permeability) and holes for the gas-bag surface and/or subsurfaces. | Integer > 0 | No porosity                |  |
| INFID      | Number of a set of ABINFL entries, that defines the one or more inflators on subsurface(s) of the GBAG surface.                    | Integer > 0 | No inflators               |  |
| HTRID      | Number of a set of HEATLOS entries, that defines the heat transfer definitions for the gas-bag surface and/or subsurfaces.         | I > 0       | No heat transfer           |  |
| INTID      | ID of an INITGAS entry specifying the initial gas composition for this GBAG.   | I > 0       | No initial gas composition |  |

| Descriptor | Meaning  |           |           |  |
|------------|--|-----------|-----------|--|
| CPGAS      | The variation of the specific heat constant at constant pressure.<br><br>CONSTANT The specific heat is constant and specified in CPGASV.   | Character | CONSTANT  |  |
| CPGASV     | The specific heat of the gas.  | Real      | Required  |  |
| RGAS       | Gas constant of the inflowing gas.   | Real      | Required  |  |
| PENV       | Environmental pressure surrounding the gas bag.  | Real      | Required  |  |
| REVERSE    | Normal auto-reverse switch.<br><br>ON The normals of the surface are automatically reversed if necessary so that they point in the same direction and provide a positive volume.<br><br>OFF The normals are not automatically reversed.                          | C         | ON        |  |
| CHECK      | Normal checking switch.<br><br>ON The normals of the surface are checked to see if they all point in the same direction and provide a positive volume.<br><br>OFF The normals are not checked.<br><br>If REVERSE is set to ON, CHECK is automatically set to ON. | C         | ON        |  |
| PINIT      | Initial pressure inside the gas bag.   | Real      | PENV      |  |
| TINIT      | Initial temperature inside the gas bag.<br>See Remark 4.   | Real      | Required. |  |
| TENV       | Environmental Temperature.   | Real > 0  | TINIT     |  |

**Remarks:**

1. The BSURF entry referenced by the BSID field must form a closed volume.
2. The pressure in the gas bag is applied to all the faces of the outer boundary.
3. TINIT is the temperature of the gas inside the volume at *time* = 0. At *time* = 0, the mass of the gas inside the gas bag is calculated as

$$m = \frac{P_{init}V}{RT_{init}}$$

where,  $P_{init}$  the initial pressure,  $V$  the volume,  $R$  the gas constant, and  $T_{init}$  the initial gas temperature.

4. The flow through exhaust openings, leakage areas and user-specified outflow rate is accumulated. The volumetric porosity contributes to the outflow of gas as

$$\dot{m}_{out} = \rho \cdot Q = \frac{p}{R \cdot T} \cdot Q$$

where

$Q$  = volumetric flow rate

$\rho$  = density inside the bag

$p$  = pressure inside the bag

$R$  = gas constant

$T$  = temperature inside the bag

$\dot{m}_{out}$  = mass outflow rate

The value of  $Q$  can be specified as a constant, as a function of the pressure difference, or as a function of time. Negative values for the volumetric flow rate are not allowed, since this would mean inflow of outside air.

5. A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.

**GBAGCOU****General Coupling to Gas-Bag Switch**

Defines a switch from full gas dynamics to uniform pressure formulation. Used in SOL 700 only.

**Format:**

| 1       | 2  | 3   | 4   | 5      | 6       | 7 | 8 | 9 | 10 |
|---------|----|-----|-----|--------|---------|---|---|---|----|
| GBAGCOU | ID | CID | GID | TSTART | PERCENT |   |   |   |    |

**Examples:**

|         |   |     |     |     |   |  |  |  |  |
|---------|---|-----|-----|-----|---|--|--|--|--|
| GBAGCOU | 1 | 100 | 101 | 0.0 | 5 |  |  |  |  |
|---------|---|-----|-----|-----|---|--|--|--|--|

| Descriptor | Meaning  |               |          |  |
|------------|--|---------------|----------|--|
| ID         | Unique number of a GBAGCOU entry.  | I > 0         | Required |  |
| CID        | Number of a COUPLE entry.  | I > 0         | Required |  |
| GID        | Number of a GBAG entry.  | I > 0         | 0.0      |  |
| TSTART     | Time after which the coupling algorithm checks if a switch to the uniform pressure method is valid. It is valid when the following is true:                              | Real $\geq 0$ | 0.0      |  |
|            | $\text{Max}\left[\frac{P_{\max} + P_{\text{average}}}{P_{\text{average}}}, \frac{P_{\text{average}} - P_{\min}}{P_{\text{average}}}\right] < \frac{\text{PERCENT}}{100}$ |               |          |  |
|            | where  |               |          |  |
|            | Pmax = maximum Eulerian pressure exerted on the SURFACE  |               |          |  |
|            | Pmin = minimum Eulerian pressure exerted on the SURFACE  |               |          |  |
|            | Paverage = average Eulerian pressure exerted on the SURFACE  |               |          |  |
| PERCENT    | Value used in validity check as defined previously.  | R $\geq 0$ .  | 0.0      |  |

**Remarks:**

1. The BSID referenced by the COUPLE entry CID and by the GBAG entry GID must be the same.
2. All Eulerian and general coupling calculations are deactivated after transition from gas dynamics to uniform pressure.

**GENEL****General Element**

Defines a general element.

**Format:**

| 1     | 2   | 3   | 4   | 5   | 6      | 7   | 8   | 9   | 10 |
|-------|-----|-----|-----|-----|--------|-----|-----|-----|----|
| GENEL | EID |     | UI1 | CI1 | UI2    | CI2 | UI3 | CI3 |    |
|       | UI4 | CI4 | UI5 | CI5 | -etc.- |     |     |     |    |

UIm -- The last item in the UI list will appear in one of fields 2, 4, 6, or 8.

|  |      |  |     |     |     |     |        |  |  |
|--|------|--|-----|-----|-----|-----|--------|--|--|
|  | "UD" |  | UD1 | CD1 | UD2 | CD2 | -etc.- |  |  |
|--|------|--|-----|-----|-----|-----|--------|--|--|

UDn -- The last item in the UD list will appear in one of fields 2, 4, 6, or 8.

|  |            |      |      |      |        |      |      |  |  |
|--|------------|------|------|------|--------|------|------|--|--|
|  | "K" or "Z" | KZ11 | KZ21 | KZ31 | -etc.- | KZ22 | KZ32 |  |  |
|  | -etc.-     |      | KZ33 | KZ43 | -etc.- |      |      |  |  |

KZmm -- The last item in the K or Z matrix will appear in one of fields 2 through 9.

|  |     |     |     |        |  |     |        |  |  |
|--|-----|-----|-----|--------|--|-----|--------|--|--|
|  | "S" | S11 | S12 | -etc.- |  | S21 | -etc.- |  |  |
|--|-----|-----|-----|--------|--|-----|--------|--|--|

Smn -- The last item in the S matrix will appear in one of fields 2 through 9.

**Example:**

|       |     |     |      |     |     |     |     |     |  |
|-------|-----|-----|------|-----|-----|-----|-----|-----|--|
| GENEL | 629 |     | 1    | 1   | 13  | 4   | 42  | 0   |  |
|       | 24  | 2   |      |     |     |     |     |     |  |
|       | UD  |     | 6    | 2   | 33  | 0   |     |     |  |
|       | Z   | 1.0 | 2.0  | 3.0 | 4.0 | 5.0 | 6.0 | 7.0 |  |
|       | 8.0 | 9.0 | 10.0 |     |     |     |     |     |  |
|       | S   | 1.5 | 2.5  | 3.5 | 4.5 | 5.5 | 6.5 | 7.5 |  |
|       | 8.5 |     |      |     |     |     |     |     |  |

| Descriptor           | Meaning  |
|----------------------|--|
| EID                  | Unique element identification number. (Integer > 0)  |
| Uli, Cli<br>UDj, CDj | Identification numbers of degrees-of-freedom in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. UIi and UDi are grid point numbers, and CLI and CDj are the component numbers. If a scalar point is given, the component number is zero. (Integer $\geq 0$ ) |
| KZij                 | Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list. (Real)   |

| Descriptor              | Meaning  |
|-------------------------|--|
| Sij                     | Values of the [S] matrix ordered by rows according to the UD list. (Real)                                    |
| "UD", "K", "Z", and "S" | Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices. |

**Remarks:**

1. The stiffness approach:

$$\begin{bmatrix} f_i \\ f_d \end{bmatrix} = \begin{bmatrix} K & -KS \\ -S^T K & S^T K S \end{bmatrix} \begin{bmatrix} u_i \\ u_d \end{bmatrix}$$

The flexibility approach:

$$\begin{bmatrix} u_i \\ f_d \end{bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{bmatrix} f_i \\ u_d \end{bmatrix}$$

where

$$\{u_i\} = [u_{i1}, u_{i2}, \dots, u_{im}]^T$$

$$\{u_d\} = [u_{d1}, u_{d2}, \dots, u_{dn}]^T$$

$$\begin{aligned} [KZ] &= \\ [K] \text{ or } [Z] &= \begin{bmatrix} KZ_{11} & \dots & \dots & \dots \\ KZ_{21} & KZ_{22} & \dots & \dots \\ KZ_{31} & KZ_{32} & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ] \end{aligned}$$

$$\begin{bmatrix} S \end{bmatrix} = \begin{bmatrix} S_{11} & \dots & S_{1n} \\ S_{21} & \dots & \dots \\ S_{31} & \dots & \dots \\ \vdots & \vdots & \vdots \\ S'_{m1} & \dots & S_{mn} \end{bmatrix}$$

The required input is the  $\{u_i\}$  list and the lower triangular portion of  $[K]$  or  $[Z]$ . Additional input may include the  $\{u_d\}$  list and  $[S]$ . If  $[S]$  is input,  $\{u_d\}$  must also be input. If  $\{u_d\}$  is input but  $[S]$  is omitted,  $[S]$  is internally calculated. In this case,  $\{u_d\}$  must contain six and only six degrees-of-freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body with rigid body motions that are defined by  $\{u_i\} = [S]\{u_d\}$ . See [General Element Capability \(GENEL\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion.

2. When the stiffness matrix  $K$  is input, the number of significant digits should be the same for all terms.
3. Double-field format may be used for input of  $K$  or  $Z$ .
4. The DMIG entry or the INPUTT4 module offer alternative methods for inputting large matrices.
5. The general element entry in the example above defines the following:

$$\{u_i\} = [1-1, 13-4, 42, 24-2]^T$$

$$\{u_d\} = [6-2, 33]^T$$

where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \quad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

**GENUDS****User Data for Notify User Defined Service or Subroutine**

The GENUDS defines integer, real and character type data that will be passed to the notify method in runtime info interface. The notify method will be called at start of load case, start of increment, end of increment and end of load case.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6       | 7      | 8      | 9      | 10 |
|--------|--------|--------|--------|--------|---------|--------|--------|--------|----|
| GENUDS | SRV_ID |        |        |        |         |        |        |        |    |
|        | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4  | IDATA5 | IDATA6 | IDATA7 |    |
|        |        | IDATA8 | IDATA9 | ...    | IDATAAn |        |        |        |    |
|        | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4  | RDATA5 | RDATA6 | RDATA7 |    |
|        |        | RDATA8 | RDATA9 | ...    | RDATAAn |        |        |        |    |
|        | “CHAR” | CDATA1 | CDATA2 | ...    | CDATAAn |        |        |        |    |
|        |        |        |        |        |         |        |        |        |    |

**Example:**

|        |        |   |   |     |  |  |  |  |  |
|--------|--------|---|---|-----|--|--|--|--|--|
| GENUDS | MY_SRV |   |   |     |  |  |  |  |  |
|        | INT    | 1 | 2 | 100 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SRV_ID     | The service identifier used in the CONNECT SERVICE statement. (Character, no default) |
| “INT”      | Keyword indicating that the following data is integer. (Character)                    |
| IDATAi     | User supplied integer data. (Integer, no default)                                     |
| “REAL”     | Keyword indicating the following data is real. (Character)                            |
| RDATAi     | User supplied real data. (Real, no default)   |
| “CHAR”     | Keyword indicating the following data is character. (Character)                       |
| CDATAi     | User supplied character data. (Character, no default)                                 |

**Remarks:**

1. The SRV\_ID is the service identifier of SCA service in the CONNECT SERVICE statement. The SCA service should have implemented the RuntimeInfo interface.
2. A CDATAi entry cannot be the character “INT”, “REAL” or “CHAR”.

**GMBC****General Enforced Displacement Definition**

Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

**Format:**

| 1    | 2      | 3      | 4      | 5      | 6  | 7      | 8      | 9      | 10 |
|------|--------|--------|--------|--------|----|--------|--------|--------|----|
| GMBC | LID    | SPCID  | C      | ENTITY | ID | METHOD | FIELD1 | FIELD2 |    |
|      | FIELD3 | FIELD4 | -etc.- |        |    |        |        |        |    |

**Example:**

|      |     |   |   |        |   |      |    |    |  |
|------|-----|---|---|--------|---|------|----|----|--|
| GMBC | 127 | 1 | 2 | GMCURV | 1 | QUAD | 1. | 2. |  |
|      | 1.0 |   |   |        |   |      |    |    |  |

| Descriptor | Meaning  |                                |          |
|------------|--|--------------------------------|----------|
| LID        | LOAD set identification number.<br>See Remark 2.   | Integer $\geq 0$               | Required |
| SPCID      | SPC set identification number. See Remark 2.   | Integer $> 0$                  | Required |
| C          | Component number in the output coordinate system (global). See Remarks 3. and 4.                                       | $0 \leq \text{Integer} \leq 6$ | Required |
| ENTITY     | Entity that the enforced displacement is applied to (Specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF).<br>See Remarks. | Character                      | Required |
| ID         | ID of the entity selected above. See Remarks.  | Integer $> 0$                  | Required |
| METHOD     | Method used to supply the data (EQUATION or TABLE, CONSTANT, LINEAR, QUAD, CUBIC). See Remarks.                        | Character                      | Remark   |
| FIELDi     | Enforced displacement data. See Remarks.   | Integer or Real                | Required |

**Remarks:**

1. GMBC is the recommended entry for specifying boundary conditions and must be selected with Case Control command SPC = SPCID.
2. LID and SPCID refer to Case Control commands for specifying loads and boundary conditions. Whenever there are several nonzero enforced motion vectors supplied, the most efficient processing of the data (single decomposition of the stiffness matrix) is accomplished by specifying both LID and SPCID.

| LID | Result   |
|-----|--|
| > 0 | Generate SPC entries with zero displacements and SPCD entries with non-zero displacements. |
| 0   | Generate SPC entries with non-zero displacements only                                      |

3. The components of motion specified by C (field 4) of all degrees-of-freedom in the output coordinate system (Global System) associated with an entity will be constrained.
4. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set. In this case the method field is not required.
5. The component is a single integer (1 or 2 or 3 etc.). Use multiple GMBC entries to enforce constraints on multiple components.
6. If METHOD = “EQUATION”, “TABLE”, or “CONSTANT” then FIELD1 is:

| METHOD   | FIELD1  | Type        |
|----------|---|-------------|
| EQUATION | ID of a DEQATN entry defining the displacement value as a function of location  | Integer > 0 |
| TABLE    | ID of a TABLE3D entry defining the displacement value as a function of location | Integer > 0 |
| CONSTANT | Value of enforced displacement  | Real        |

7. When METHOD = CONSTANT, a constant displacement is specified for the FEEDGE, GMCURV, FEFACE, and GMSURF entities.
8. If ENTITY = “FEEDGE” the METHOD field can be used to specify, linear, quadratic, and cubic displacements. FIELD1 and FIELD2 correspond to GRID1 and GRID2 on the FEEDGE entry. The values in FIELD3 and FIELD4 are:

| Applying Linear, Quadratic, and Cubic Displacements to an FEEDGE |                           |                           |
|--|---------------------------|---------------------------|
| METHOD   | FIELD3                    | FIELD4                    |
| LINEAR   | blank                     | blank                     |
| QUAD   | Value at 1/2 chord length | blank                     |
| CUBIC  | Value at 1/3 chord length | Value at 2/3 chord length |

9. If ENTITY = “FEFACE” the METHOD field specifies linear or quadratic displacements. The values of FIELDi are location specific:
  - Quadrilateral FEFACE

| METHOD | FIELD1 through FIELD4   | FIELD5 through FIELD8               | FIELD9                    | Displacement Function |
|--------|-------------------------|-------------------------------------|---------------------------|-----------------------|
| LINEAR | Value at GRID1, 2, 3, 4 | blank                               | blank                     | Linear                |
| QUAD   | Value at GRID1, 2, 3, 4 | Value at mid side of EDGE1, 2, 3, 4 | Value at middle of FEFACE | Quadratic             |

- Triangular FEFACE

| METHOD | FIELD1 through FIELD3 | FIELD4 through FIELD6            | Displacement Function |
|--------|-----------------------|----------------------------------|-----------------------|
| LINEAR | Value at GRID1, 2, 3  | blank                            | Linear                |
| QUAD   | Value at GRID1, 2, 3  | Value at mid side of EDGE1, 2, 3 | Quadratic             |

10. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is: GRIDs followed by FEEDGEs followed by GMCURVs followed by FEFACEs followed by GMSURFs. This means that:
- In general the program does not allow the user to supply multiple values of enforced displacements for the same component (C).
  - Displacement values specified for each component of a given GMSURF entry are applied to the same component of all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the GMSURF.
  - Displacement values specified for each component of different GMSURF entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple GMSURFs.
  - Displacement values specified for a given FEFACE entry are applied to all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the FEFACE. This data overrides the data that is specified for all the components of the given GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the FEFACE by using GMSURF entries.
  - Displacement values specified for each component of different FEFACE entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple FEFACES. This data overrides the data that is specified for all the components of the given FEEDGE and GRIDs by using GMSURF entries.
  - Displacement values specified for each component of a given GMCURV entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the GMCURV. This data overrides the data for all the components that are specified for the given FEEDGE and GRIDs by using GMSURF or FEFACE entries.

- Displacement values specified for each component of different GMCURV entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple GMCURVs. This data overrides the data for all the components that are specified for the given GRIDs by using GMSURF or FEFACE entries.
- Displacement values specified for each component of a given FEEDGE entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the FEEDGE. This data overrides the data for all the components that is specified for the given FEEDGE and GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Displacement values specified for each component of different FEEDGE entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple FEEDGES. This data overrides the data for all the components that are specified for the given GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Grids have the highest priority, i.e., any value/property specified using a GRID entry overrides all other information associated with that GRID. If multiple entries are used for a given GRID, e.g., multiple SPCs, then the existing rules govern (SPCs are combined, FORCE is added, SPCDs for the same component are not allowed).
- It is important to recall that these displacements are assumed to be in the Global Coordinate System and that the interconsistency of the output coordinate systems of the various GRIDs, FEEDGES, FEFACEs is not checked.
- If an entity is specified on both a GMBC and GMSPC entry then the GMSPC specification will be ignored.
- Coordinate transformations should not be applied to grid points on FEEDGE, GMCURVE, FEFACE or GMSURF.

11. For the example in [Figure 9-96](#),

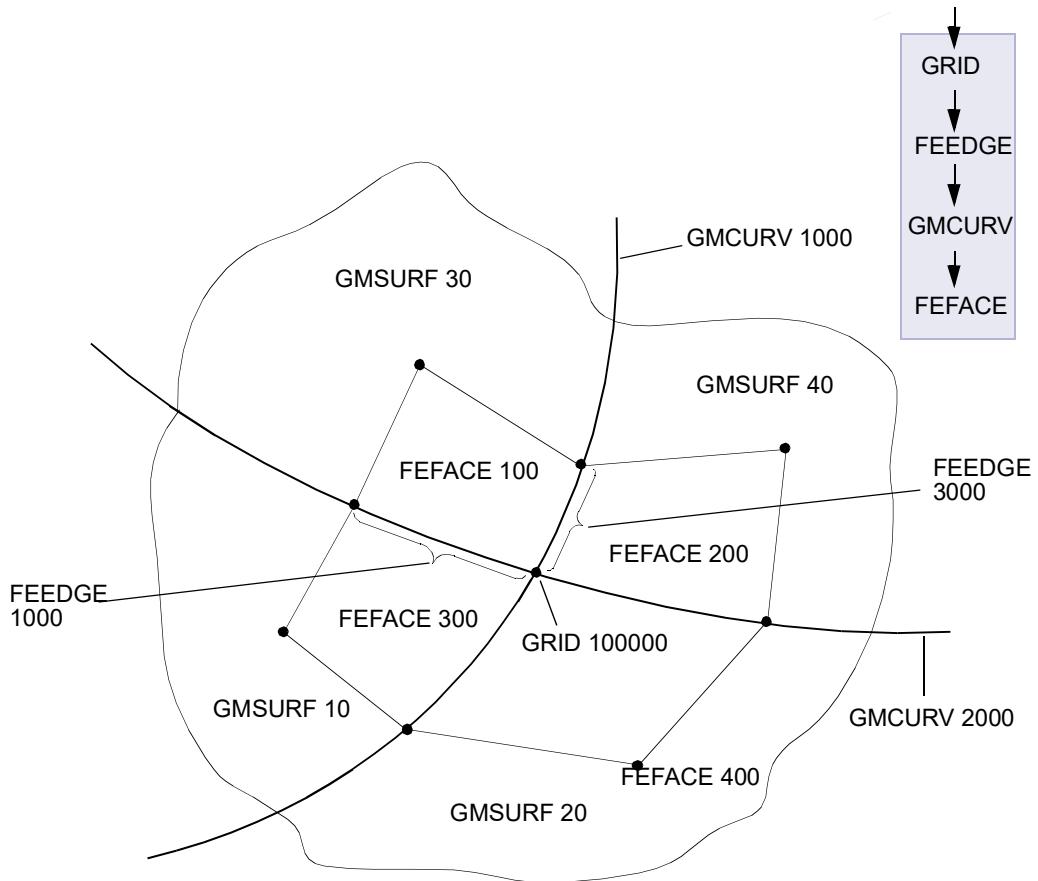


Figure 9-96 Use of Multiple Surface and Curves

- The enforced displacement for GRID 100000 can be specified using SPCD, GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 18](#) describes the outcome of using these different methods:

**Table 18** Enforced Displacement Used for GRID 10000

| When Specified Using   | Action   |
|------------------------|--|
| SPCD                   | Overrides all other information supplied for all components.   |
| Single GMBC (FEEDGE)   | Overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries.   |
| Multiple GMBC (FEEDGE) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries. |
| Single GMBC (GMCURV)   | Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.  |
| Multiple GMBC (GMCURV) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.              |
| Single GMBC (FEFACE)   | Overrides information supplied for all components using GMBC(GMSURF) entries.  |
| Multiple GMBC (FEFACE) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.                            |
| Single GMBC (GMSURF)   | Values are applied.  |
| Multiple GMBC (GMSURF) | Values are averaged on a component basis.  |

- The enforced displacement for GRID DOFs and edge DOFs belonging to FEEDGE 10000 can be specified using GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 19](#) describes the outcome of using these different methods:

**Table 19** Enforced Displacement Used for FEEDGE 10000

| When Specified Using      | Action  |
|---------------------------|---|
| Single GMBC<br>(FEEDGE)   | Overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries.   |
| Multiple GMBC<br>(FEEDGE) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries. |
| Single GMBC<br>(GMCURV)   | Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.   |
| Multiple GMBC<br>(GMCURV) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.               |
| Single GMBC<br>(FEFACE)   | Overrides information supplied for all components using GMBC(GMSURF) entries.   |
| Multiple GMBC<br>(FEFACE) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.                             |
| Single GMBC<br>(GMSURF)   | Values are applied.   |
| Multiple GMBC<br>(GMSURF) | Values are averaged on a component basis.   |

- The enforced displacement for the GRID DOFs and edge DOFs belonging to GMCURV 1000 can be specified using GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. [Table 20](#) describes the outcome of using these different methods:

**Table 20** Enforced Displacement Used for GMCURV 10000

| When Specified Using   | Action  |
|------------------------|---|
| Single GMBC (GMCURV)   | Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.   |
| Single GMBC (FEFACE)   | Overrides information supplied for all components using GMBC(GMSURF) entries.   |
| Multiple GMBC (FEFACE) | Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries. |
| Single GMBC (GMSURF)   | Values are applied.   |
| Multiple GMBC (GMSURF) | Values are averaged on a component basis.   |

- The enforced displacement for the GRID DOFs, the edge DOFs, and the face DOFs belonging to FEFACE 300 can be specified using GMBC referring to a FEFACE and GMBC referring to a GMSURF. [Table 21](#) describes the outcome of using these different methods:

**Table 21** Enforced Displacement Used for FEFACE 300

| When Specified Using | Action  |
|----------------------|---|
| Single GMBC (FEFACE) | Overrides information supplied for all components using GMBC(GMSURF) entries. |
| Single GMBC (GMSURF) | Values are applied.   |

**GMBNDC****Geometric Boundary - Curve**

Defines a geometric boundary consisting of either h- or p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or solid elements.

**Format:**

| 1      | 2      | 3     | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|--------|-------|--------|-----|-----|-----|-----|-----|----|
| GMBNDC | BID    | GRIDI | GRIDF  |     |     |     |     |     |    |
|        | ENTITY | ID1   | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|        |        | ID8   | -etc.- |     |     |     |     |     |    |

**Examples:**

|        |        |     |     |  |  |  |  |  |  |
|--------|--------|-----|-----|--|--|--|--|--|--|
| GMBNDC | 1      | 101 | 106 |  |  |  |  |  |  |
|        | GMCURV | 1   |     |  |  |  |  |  |  |

|        |        |     |     |    |    |    |  |  |  |
|--------|--------|-----|-----|----|----|----|--|--|--|
| GMBNDC | 1      | 101 | 106 |    |    |    |  |  |  |
|        | FEEDGE | 11  | 12  | 13 | 14 | 15 |  |  |  |

|        |      |     |     |     |     |  |  |  |  |
|--------|------|-----|-----|-----|-----|--|--|--|--|
| GMBNDC | 1    | 101 | 106 |     |     |  |  |  |  |
|        | GRID | 102 | 103 | 104 | 105 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Boundary identification number to be referenced by a GMINTC entry. (Integer > 0)                          |
| GRIDI      | Initial grid identification number for boundary. (Integer > 0)  |
| GRIDF      | Final grid identification number for boundary. (Integer > 0)  |
| ENTITY     | Entity type for defining boundary. (Character)  |
| IDi        | Entity identification numbers for boundary of subdomain. Values in the list must be unique. (Integer > 0) |

**Remarks:**

1. All boundary identification numbers must be unique.
2. GRIDI and GRIDF define the end points of the boundary.
3. For each boundary, one of the entity types GMCURV, FEEDGE, or GRID is required.
4. For the GMCURV entity type, if there are multiple paths on the GMCURV from the GRIDI to the GRIDF, such as two segments of a circle, the FEEDGE or GRID method must be used instead to uniquely define the path.
5. For the GRID entity type, the entities should be listed in order from the GRIDI to the GRIDF. The GRIDI and GRIDF need not be repeated in the IDi list.

6. If more than one boundary references the same GMCURV entry with the same GRIDI and GRIDF, then the FEEDGE or GRID entity type must be used instead for each to uniquely identify the boundaries.
7. Multiple continuation entries may be specified for additional entity identification numbers, IDi.

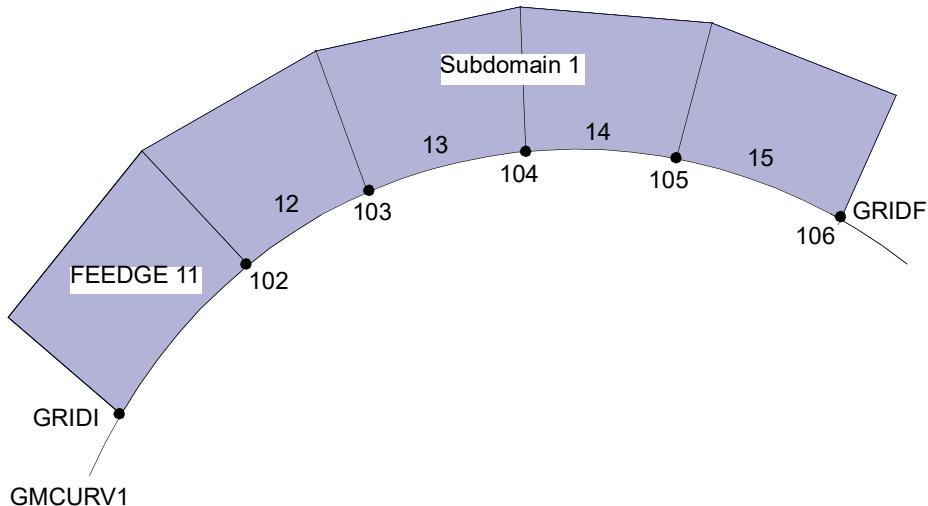


Figure 9-97 Geometric Boundary Definition

8. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.
9. When it is referenced by Bulk Data entry, CINTC, which defines an h-interface element, the value of field ENTITY must be GRID.

**GMBNDS****Geometric Boundary - Surface**

Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.

**Format:**

| 1      | 2      | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|--------|-----|--------|-----|-----|-----|-----|-----|----|
| GMBNDS | BID    |     |        |     |     |     |     |     |    |
|        | ENTITY | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|        |        | ID8 | -etc.- |     |     |     |     |     |    |

**Examples:**

|        |        |   |  |  |  |  |  |  |  |
|--------|--------|---|--|--|--|--|--|--|--|
| GMBNDS | 1      |   |  |  |  |  |  |  |  |
|        | GMSURF | 1 |  |  |  |  |  |  |  |

|        |        |    |    |    |    |    |    |  |  |
|--------|--------|----|----|----|----|----|----|--|--|
| GMBNDS | 1      |    |    |    |    |    |    |  |  |
|        | FEFACE | 11 | 12 | 13 | 14 | 15 | 16 |  |  |

|        |      |     |     |     |     |     |     |     |  |
|--------|------|-----|-----|-----|-----|-----|-----|-----|--|
| GMBNDS | 1    |     |     |     |     |     |     |     |  |
|        | GRID | 101 | 102 | 103 | 104 | 105 | 106 | 107 |  |
|        |      | 108 | 109 | 110 | 111 | 112 |     |     |  |

**Descriptor**      **Meaning**

|        |                                    |             |          |
|--------|------------------------------------|-------------|----------|
| BID    | Boundary identification number.    | Integer > 0 | Required |
| ENTITY | Entity type for defining boundary. | Character   | Required |
| IDi    | Entity ID i for boundary           | Integer > 0 | Optional |

**Remarks:**

1. All BIDs must be unique.
2. For each boundary, one of the entity types GMSURF, FEFACE, or GRID is required.
3. For the GMSURF entity type, all the faces referencing the GMSURF will be included in the boundary.
4. If more than one boundary references the same GMSURF, then the FEFACE or GRID entity type must be used instead for each to uniquely identify the boundaries.
5. Multiple continuation entries may be used without repeating the ENTITY field.

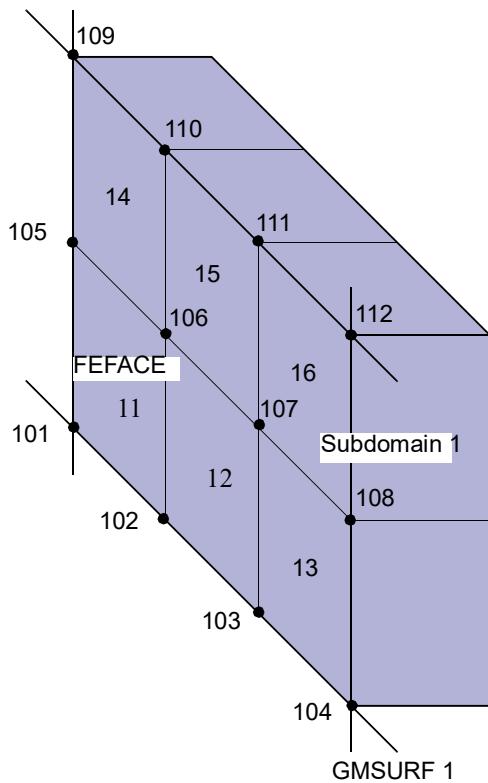


Figure 9-98 Surface Boundary Definition

6. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.

**GMCONV**

## Define Convection Boundary Conditions

Defines free convection boundary conditions for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

**Format:**

| 1      | 2   | 3      | 4  | 5      | 6      | 7      | 8 | 9 | 10 |
|--------|-----|--------|----|--------|--------|--------|---|---|----|
| GMCONV | LID | ENTITY | ID | METHOD | FIELD1 | FIELD2 |   |   |    |

**Example:**

|        |    |        |   |    |      |      |  |  |  |
|--------|----|--------|---|----|------|------|--|--|--|
| GMCONV | 15 | FEFACE | 3 | 10 | 1001 | 20.0 |  |  |  |
|--------|----|--------|---|----|------|------|--|--|--|

**Descriptor      Meaning**

|        |   |                  |          |
|--------|---|------------------|----------|
| LID    | Load set identification number.   | Integer > 0      | Required |
| ENTITY | Entity that the convection boundary condition is applied to (specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remarks. | Character        | Required |
| ID     | ID of the entity selected above. See Remarks.   | Character        | Required |
| METHOD | Method used to specify the data. See Remark 2.  | Integer $\geq 0$ | 0        |
| FIELD1 | Convection heat transfer coefficient data. See Remark 2.  | Integer or Real  | Required |
| FIELD2 | Ambient temperature data. See Remark 2.   | Integer or Real  | Required |

**Remarks:**

1. For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
2. METHOD specifies the data types of FIELD1 and FIELD2 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

| METHOD | FIELD1  | FIELD2  |
|--------|---|---|
| 0      | Value of heat transfer coefficient (Real > 0.0).  | Value of ambient temperature (Real).  |
| 1      | Value of heat transfer coefficient (Real > 0.0).  | ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).  |
| 2      | Value of heat transfer coefficient (Real > 0.0).  | ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0). |
| 10     | ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).  | Value of ambient temperature (Real).  |
| 11     | ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).  | ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).  |
| 12     | ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).  | ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0). |
| 20     | ID of a TABLE3D entry defining the heat transfer coefficient a location (Integer > 0).                | Value of ambient temperature (Real).  |
| 21     | ID of a TABLED3 entry defining the heat transfer coefficient as a function of location (Integer > 0). | ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).  |
| 22     | ID of a TABLE3D entry defining the heat transfer coefficient as a function of location (Integer > 0). | ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0). |

3. The proper units must be specified for the value of FIELD1.

| Units of FIELDi for Different ENTITY Fields |                     |
|---|---------------------|
| ENTITY                                      | Units               |
| GRID  | Power/Degree        |
| FEEDGE                                      | Power/Length-Degree |
| GMCURV                                      | Power/Length-Degree |
| FEFACE                                      | Power/Area-Degree   |
| GMSURF                                      | Power/Area-Degree   |

4. Multiple values of convection boundary conditions can be applied to the same geometry region. In general, a hierarchy is set to resolve the conflicts arising in the input data:
  - a. Information provided on multiple GMSURF and FEFACE entries are added for all FEFACE entries.
  - b. Information provided on multiple GMCURV and FEEDGE entries are added for all FEEDGE entries.

**GMCORD**

## Convective/Follower Coordinate System Definition

Defines a convective/follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.

**Format:**

| 1      | 2   | 3      | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|-----|--------|-----|-----|---|---|---|---|----|
| GMCORD | CID | ENTITY | ID1 | ID2 |   |   |   |   |    |

**Example:**

|        |     |        |    |    |  |  |  |  |  |
|--------|-----|--------|----|----|--|--|--|--|--|
| GMCORD | 101 | GMCURV | 26 | 44 |  |  |  |  |  |
|--------|-----|--------|----|----|--|--|--|--|--|

| Descriptor | Meaning   |             |          |  |
|------------|---|-------------|----------|--|
| CID        | Coordinate system identification number, unique with respect to all CORDij entries. | Integer > 0 | Required |  |
| ENTITY     | Type of Bulk Data entry that is used to define the coordinate system. See Remark 3. | Character   | Required |  |
| ID1, ID2   | Entity identification numbers. See Remark 3.  | Integer > 0 | Required |  |

**Remarks:**

1. GMCORD defines a (convective) coordinate system associated with an entity. This type of coordinate system can be used to apply loads and boundary conditions only.
2. GMCORD can only be specified for p-version elements.
3. The Bulk Data entries referenced by ID1 and ID2 depends on ENTITY.

| ENTITY | ID1             | ID2             |
|--------|-----------------|-----------------|
| FEEDGE | FEEDGE entry ID | FEFACE entry ID |
| GMCURV | GMCURV entry ID | GMSURF entry ID |
| FEFACE | FEFACE entry ID | Blank           |
| GMSURF | GMSURF entry ID | Blank           |

- For ENTITY = “FEEDGE” normal is defined by the FEFACE.
- For ENTITY = “GMCURV” normal is defined by the GMSURF.

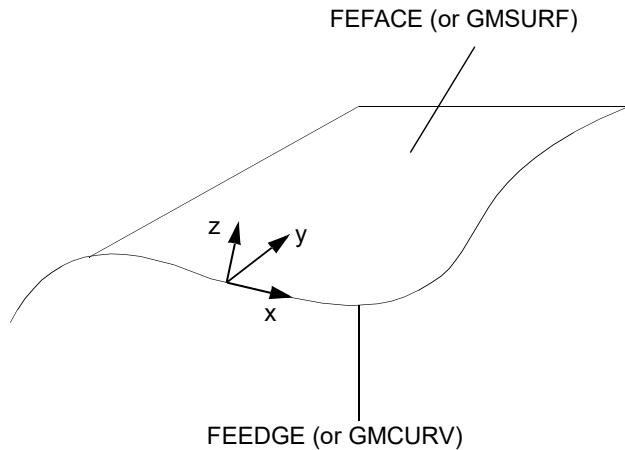


Figure 9-99

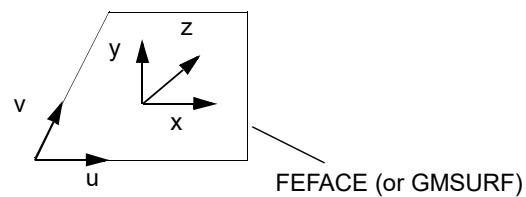


Figure 9-100

**GMCURV****Curve Definition**

Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.

Format:

| 1      | 2         | 3        | 4     | 5     | 6      | 7 | 8 | 9 | 10 |
|--------|-----------|----------|-------|-------|--------|---|---|---|----|
| GMCURV | CURVID    | GROUP    | CIDIN | CIDBC |        |   |   |   |    |
|        | Evaluator | Specific | Data  | and   | Format |   |   |   |    |

Example:

|        |          |        |     |     |     |     |         |  |  |
|--------|----------|--------|-----|-----|-----|-----|---------|--|--|
| GMCURV | 101      | FENDER |     |     |     |     |         |  |  |
|        | RPC      | POINT  |     |     |     |     |         |  |  |
|        | 0.0, 2.0 | 1.0    | 1.0 |     | 2.0 | 3.0 | 4.0,1.0 |  |  |
|        | 0.0, 2.0 | 1.0    | 1.0 | 1.0 | 2.0 | 3.0 | 4.0     |  |  |

| Descriptor | Meaning  |                  |  |          |
|------------|--|------------------|--|----------|
| CURVID     | Unique identification number. See Remarks 1. and 2.  | Integer > 0      |  | Required |
| GROUP      | Group of curves/surfaces that this curve belongs to. See Remarks 4. through 8.   | Character        |  | Required |
| CIDIN      | Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular. | Integer $\geq 0$ |  | 0        |
| CIDBC      | Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.           | Integer $\geq 0$ |  | 0        |

#### Remarks:

1. GMCURV is used to calculate geometric information only. The edges of the finite elements that are connected to the curve will be parametric cubic curves that are calculated from the more complex curve geometry.
2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
4. The GROUP parameter is initialized by an CONNECT GEOMEVAL statement in the FMS Section. This command specifies the evaluator that will be used for this curve.
5. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized on the CONNECT FMS statement.

6. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 4 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

| 1 | 2     | 3      | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---|-------|--------|-------|-------|-------|-------|-------|-------|----|
|   | RPC   | REPRES |       |       |       |       |       |       |    |
|   | XW(1) | XW(2)  | XW(3) | XW(4) | YW(1) | YW(2) | YW(3) | YW(4) |    |
|   | ZW(1) | ZW(2)  | ZW(3) | ZW(4) | W(1)  | W(2)  | W(3)  | W(4)  |    |

| Descriptor               | Meaning   | Character | Required |
|--------------------------|---|-----------|----------|
| RPC                      | Rational Parametric Cubic Curve.                              | Character | Required |
| REPRES                   | Representation of the curve (“ALGEBRAIC”, “POINT”, “BEZIER”). | Character | Required |
| XW(1)<br>through<br>W(4) | Data used to define the curve.                                | Real      | Required |

- A rational parametric curve (RPC) is defined as

$$x(t) = \frac{xw(t)}{w(t)}$$

$$y(t) = \frac{yw(t)}{w(t)}$$

$$z(t) = \frac{zw(t)}{w(t)}$$

$$0.0 \leq t \leq 1.0$$

- For REPRES = “ALGEBRAIC”, the parametric curve is defined by the algebraic coefficients ( $a, b, c, d$ ) for a rational cubic equation.

$$P(t) = at^3 + bt^2 + ct + d$$

Expressed in matrix form:

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} a \\ b \\ c \\ d \end{bmatrix}$$

for the Bulk Data input as algebraic coefficients

$$xw(t) = XW(1)t^3 + XW(2)t^2 + XW(3)t + XW(4)$$

$$yw(t) = YW(1)t^3 + YW(2)t^2 + YW(3)t + YW(4)$$

$$zw(t) = ZW(1)t^3 + ZW(2)t^2 + ZW(3)t + ZW(4)$$

$$w(t) = W(1)t^3 + W(2)t^2 + W(3)t + W(4)$$

where  $XW(i)$ ,  $YW(i)$ ,  $ZW(i)$ , and  $W(i)$  are the algebraic coefficients for each of the independent equations  $xw(t)$ ,  $yw(t)$ ,  $zw(t)$ , and  $w(t)$ .

and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “BEZIER”, the curve parametric is defined by four rational Bezier control points ( $V_1$ ,  $V_2$ ,  $V_3$ , and  $V_4$ ) expressed in matrix form

$$P(t) = V_1(1-t)^3 + V_23t(1-t)^2 + V_33t^2(1-t) + V_4t^3$$

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

where Bezier constants are

$$\begin{bmatrix} -1 & 3 & -3 & 1 \\ 3 & -6 & 3 & 0 \\ -3 & 3 & 0 & 0 \\ 1 & 0 & V_0 & 0 \end{bmatrix}$$

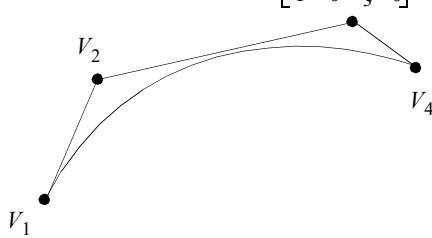


Figure 9-101 REPRES = “BEZIER”

for Bulk Data defined as Bezier control points

$$xw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$zw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

where  $XW(i)$ ,  $YW(i)$ ,  $ZW(i)$ , and  $W(i)$  correspond to  $V_p$   
and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “POINT”, the parametric curve is defined by four uniformly spaced rational points that are all on the curve similarly expressed in matrix form:

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

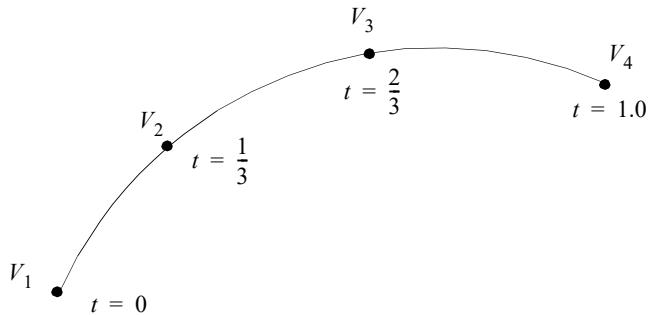


Figure 9-102 RÉPRES = “POINT”  
22EQ222

for the Bulk Data input are uniformly spaced rational points

$$xw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$zw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

and  
 $x(t) = \frac{xw(t)}{w(t)}$ ;  $y(t) = \frac{yw(t)}{w(t)}$ ;  $z(t) = \frac{zw(t)}{w(t)}$

7. If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 3 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first “field”.

| 1 | 2              | 3      | 4      | 5      | 6   | 7     | 8     | 9 | 10 |
|---|----------------|--------|--------|--------|-----|-------|-------|---|----|
|   | EQUATION, MINU | MAXU   | IDX    | IDY    | IDZ | IDDXU | IDDYU |   |    |
|   | IDDZU          | IDDXU2 | IDDYU2 | IDDZU2 |     |       |       |   |    |

| Descriptor                   | Meaning  |                  |          |  |
|------------------------------|--|------------------|----------|--|
| EQUATION                     | EQUATION method is to be used  | Character        |          |  |
| MINU, MAXU                   | Range of the curve parameter $u$ . If MAXU is found less than MINU, the range is assumed to be $[-\infty, +\infty]$ .  | Real             | 0.0,1.0  |  |
| IDX,<br>IDY,<br>IDZ          | ID of DEQATN entries providing equations for the X,Y,Z coordinates of the curve in terms of the curve parameter $u$ .  | Integer > 0      | Required |  |
| IDDXU,<br>IDDYU,<br>IDDZU    | ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the curve parameter $u$ . If a value of 0 is used, the derivatives are computed numerically.         | Integer $\geq 0$ | 0        |  |
| IDDXU2,<br>IDDYU2,<br>IDDZU2 | ID of DEQATN entries providing equations for the second derivatives of X,Y,Z functions with respect to the curve parameter $u$ . If a value of 0 is used, the second derivatives are computed numerically. | Integer > 0      | 0        |  |

8. When a user-supplied geometry evaluator is selected for this curve (through the CONNECT GEOMEVAL FMS command) the continuation entries will not be interpreted. In this case an image of this entry is passed on to the evaluator modules provided by the user. Depending on the configuration, these modules could either be linked in with Nastran or connected with Nastran during execution. If these modules are not accessible, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

- CONNECT GEOMEVAL FENDER,CATIA,’/u/kiz/data’, Version = 68 as of 1/3/94
- and the GMCURV Bulk Data entry is provided as follows:

| 1      | 2     | 3      | 4   | 5   | 6   | 7  | 8 | 9 | 10 |
|--------|-------|--------|-----|-----|-----|----|---|---|----|
| GMCURV | 102   | FENDER |     |     |     |    |   |   |    |
|        | Sweep | /u/kiz | 2.5 | arc | 2.7 | 66 |   |   |    |

- In this case, “Sweep /u/kiz 2.5 arc 2.7 66” is passed to the geometry evaluator supplied by the user, and it is expected that the user supplied routines to interpret and use this record.

**GMINTC****Geometric Interface -- Curve**

Defines an interface element along a curve interface between boundaries of multiple subdomains. Typically, the boundaries will consist of edges of p-shell subdomains but also may consist of p-beam subdomains or edges of p-solid subdomains.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| GMINTC | EID | PID | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 |    |

**Example:**

|        |      |   |   |   |  |  |  |  |  |
|--------|------|---|---|---|--|--|--|--|--|
| GMINTC | 1001 | 1 | 1 | 2 |  |  |  |  |  |
|--------|------|---|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (Integer > 0)                            |
| PID        | Property identification number of a PINTC property entry. (Integer > 0) |
| IDi        | Boundary identification number of a GMBNDC entry. (Integer > 0)         |

**Remarks:**

1. All element identification numbers must be unique.
2. For the curve interface it is recommended that only two boundaries be specified.
3. All of the end points for each boundary IDi should be coincident, and may not refer to the same grid point. The two end points of a particular boundary may not refer to the same grid, because there would be multiple directions. The boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.
4. Connecting curve boundaries of solid p-elements is not recommended because of the possibility of stress singularities.

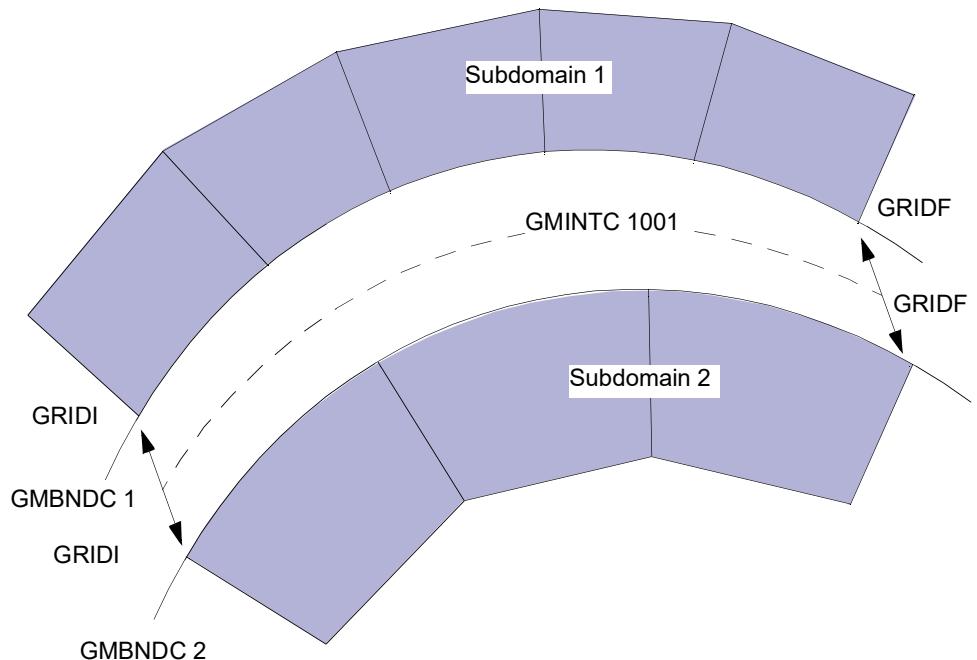


Figure 9-103 Geometric Interface Element Definition (Exploded View)

5. Because of the structure of the interface matrices, the sparse solver (default) should be used for linear statics, and the Lanczos eigensolver should be used for normal modes. In addition, for normal modes, SYSTEM(166) = 4 should be set for models where the shell normal rotations are parallel on the boundaries.

**GMINTS****Geometric Interface -- Surface**

Defines an interface element along a surface interface between boundaries of multiple subdomains. Typically, the boundaries will consist of faces of p-solid subdomains, but also may consist of p-shell subdomains.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8 | 9 | 10 |
|--------|-----|-----|-----|-----|-----|-----|---|---|----|
| GMINTS | EID | PID | ID1 | ID2 | ID3 | ID4 |   |   |    |

**Example:**

|        |      |   |   |   |  |  |  |  |  |
|--------|------|---|---|---|--|--|--|--|--|
| GMINTS | 1001 | 1 | 1 | 2 |  |  |  |  |  |
|--------|------|---|---|---|--|--|--|--|--|

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

|     |                                 |             |          |
|-----|---------------------------------|-------------|----------|
| EID | Element identification number.  | Integer > 0 | Required |
| PID | Property identification number. | Integer > 0 | Required |
| IDi | Boundary IDi of subdomain       | Integer > 0 | Required |

**Remarks:**

1. All EIDs must be unique.
2. The PID refers to a PINTS Bulk Data entry.
3. The boundary IDi of each subdomain must be defined on a GMBNDS Bulk Data entry.
4. For the surface interface, more than two boundaries are possible, but should be used carefully.
5. The perimeters of each boundary i should be coincident. In addition, the boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.

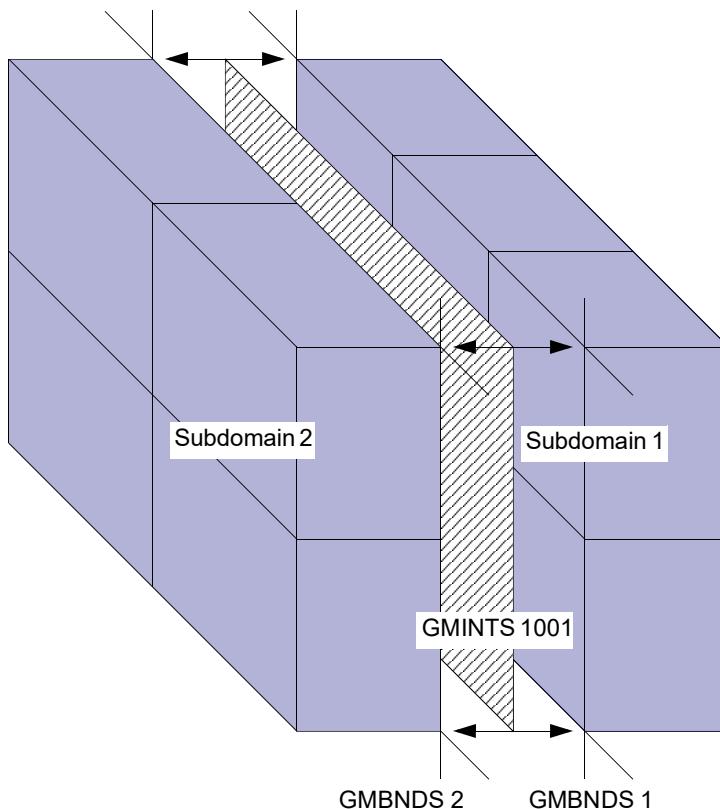


Figure 9-104 Geometric Interface Element Definition (Exploded View)

**GMLOAD****General Load Definition**

Define the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7      | 8  | 9      | 10 |
|--------|--------|--------|--------|--------|--------|--------|----|--------|----|
| GMLOAD | LID    | CID    | N1     | N2     | N3     | ENTITY | ID | METHOD |    |
|        | FIELD1 | FIELD2 | FIELD3 | FIELD4 | -etc.- |        |    |        |    |

**Example:**

|        |     |      |     |    |    |        |    |        |  |
|--------|-----|------|-----|----|----|--------|----|--------|--|
| GMLOAD | 105 | 11   | 1.0 | 0. | 0. | FEFACE | 41 | LINEAR |  |
|        | 10. | -10. | 10. |    |    |        |    |        |  |

| Descriptor | Meaning  |                   |            |  |
|------------|--|-------------------|------------|--|
| LID        | Load set identification number.  | Integer > 0       | Required   |  |
| CID        | Coordinate system in which the load is supplied. See Remark 2.   | Integer $\geq -1$ | 0          |  |
| Ni         | Direction of the force vector or axis of rotation of the moment. See Remark 3.   | Real              | 0., 0., 1. |  |
| ENTITY     | Entity that is being loaded (FEEDGE, GMCURV, FEFACE, GMSURF).  | Character         | Required   |  |
| ID         | ID of the entity selected by ENTITY.   | Integer $\geq 0$  | Required   |  |
| METHOD     | Method used to specify forces (EQUATION, TABLE, CONSTANT, LINEAR, QUAD, CUBIC) or moments (MEQUA, MTABLE, MCONST, MLINEAR, MQUAD, MCUBIC). See Remarks 4. through 6. | Character         | Required   |  |
| FIELDi     | Load magnitude data. See Remarks 4. through 8.   | Real or Integer   |            |  |

**Remarks:**

1. GMLOAD is the only method of applying forces and moments to any FEEDGE, FEFACE, GMCURV, or GMSURF in the model.
2. If CID=-1, the coordinate system on the edge or face is a local system based on the FEEDGE or FEFACE definition. (Note that an edge only has the tangent direction uniquely defined.)
3. If N1=N2=N3=0., the normal direction to the face is assumed, with the positive sense dependent on the FEFACE definition. No load will be applied for edges.

4. For both an FEEDGE and FEFACE, the METHOD field can be used to specify equation, table or constant load density. The value of FIELD1 is method-specific:

| Applying Equation, Table or Constant Load Density |  |
|---|--|
| METHOD  | FIELD1   |
| EQUATION, MQUA                                    | ID of a DEQATN entry defining the load density as a function of location.  |
| TABLE, MTABLE                                     | ID of a TABLE3D entry defining the load density as a function of location. |
| CONSTANT, MCONST                                  | Value of load density.   |

5. For an FEEDGE, the METHOD field can be used to specify linear, quadratic or cubic load density. The values of FIELDi are method-specific:

| Applying Linear, Quadratic or Cubic Load Density to an FEEDGE |                    |                    |                             |                             |              |
|---|--------------------|--------------------|-----------------------------|-----------------------------|--------------|
| METHOD  | FIELD1             | FIELD2             | FIELD3                      | FIELD4                      | Load Density |
| LINEAR,<br>MLINEAR  | Value at<br>GRID 1 | Value at<br>GRID 2 | blank                       | blank                       | Linear       |
| QUAD, MQUAD   | Value at<br>GRID 1 | Value at<br>GRID 2 | Value at 1/2 edge<br>length | blank                       | Quadratic    |
| CUBIC,<br>MCUBIC  | Value at<br>GRID 1 | Value at<br>GRID 2 | Value at 1/3 edge<br>length | Value at 2/3 edge<br>length | Cubic        |

6. For an FEFACE, the METHOD field can be used to specify linear or quadratic load density. The edges of the face are defined in the order of the grids entered (e.g., edge 1 is between the first and second grid etc.). The values of FIELDi are method-specific:

| Applying Linear and Quadratic Load Density to a Quadrilateral FEFACE |                             |                                     |                              |                 |
|--|-----------------------------|-------------------------------------|------------------------------|-----------------|
| METHOD   | FIELD1 through<br>FIELD4    | FIELD5 through<br>FIELD8            | FIELD9                       | Load<br>Density |
| LINEAR,MLINEAR   | Value at<br>GRID 1, 2, 3, 4 | blank                               | blank                        | Linear          |
| QUAD, MQUAD  | Value at<br>GRID 1, 2, 3, 4 | Value at midside of<br>EDGE 1,2,3,4 | Value at middle<br>of FEFACE | Quadratic       |

| Applying Linear and Quadratic Load Density to a Triangular FEFACE |                       |                                  |              |
|---|-----------------------|----------------------------------|--------------|
| METHOD  | FIELD1 through FIELD3 | FIELD4 through FIELD6            | Load Density |
| LINEAR,<br>MLINEAR  | Value at GRID 1, 2, 3 | blank                            | Linear       |
| QUAD, MQUAD   | Value at GRID 1, 2, 3 | Value at midside of EDGE 1, 2, 3 | Quadratic    |

7. The proper units must be specified for the value of FIELD*i*.

| Units of FIELD <i>i</i> for Different ENTITY Fields |             |
|---|-------------|
| ENTITY  | Units       |
| FEEDGE  | Load/Length |
| GMCURV  | Load/Length |
| FEFACE  | Load/Area   |
| GMSURF  | Load/Area   |

8. The load density applied to the edge or face is given by the product of the specified density with the direction vector.
9. The shell p-elements do not have stiffness in the direction of the normal rotation. Any component of moment applied in that direction will be ignored.
10. In general, a hierarchy is set to resolve the conflicts arising in the input data:
- Information provided on multiple GMSURF and FEFACE entries are added for all GRID, FEEDGE, and FEFACE degrees-of-freedom.
  - Information provided on multiple GMCURV and FEEDGE entries are added for all GRID and FEEDGE degrees-of-freedom.
  - Loads are summed over all entities.

**GMNURB****3D Contact Region Made Up of NURBS in SOL 600**

Defines a 3D contact region made up of NURBS using the MSC Marc style used in SOL 600 only.

**Format:**

| 1      | 2  | 3        | 4         | 5         | 6        | 7        | 8   | 9               | 10 |
|--------|----|----------|-----------|-----------|----------|----------|---|-----------------|----|
| GMNURB | ID | NPTU     | NPTV      | NORU      | NORV     | NSUBU    | NSUBV   | NTRIM           |    |
|        |    | G1 or X1 | G2 or Y1  | G3 or Z1  | G4 or X2 | G5 or Y2 | G6 or Z2  | G7              |    |
|        |    | G8 or X3 | G9 or Y3  | G10 or Z3 | etc.     |          | [abs(nptu)*n<br>ptv values]                                   | See<br>Remark 3 |    |
|        |    | Homo1    | Homo2     | Homo3     | Homo4    | Homo5    | Homo6   | Homo7           |    |
|        |    | Homo8    | Homo9     | Homo10    | Homo11   | etc      | [nptu*nptv<br>values]   |                 |    |
|        |    | Knot1    | Knot2     | Knot3     | Knot4    | Knot5    | Knot6   | Knot7           |    |
|        |    | Knot8    | Knot9     | Knot10    | etc.     |          | [(nptu+noru)<br>+(nptv+norv)<br>values]                       |                 |    |
|        |    | IDtrim   | NPTUtrim  | NORUtrim  | NSUBtrim |          | (repeat this<br>and all<br>following<br>lines NTRIM<br>times) |                 |    |
|        |    |          | Xisoparam | Yisoparam |          |          | (NPTUtrim<br>entries)   |                 |    |
|        |    |          | Homo1     | Homo2     | Homo3    | etc      | (NPTUtrim<br>entries)   |                 |    |
|        |    |          | Knot1     | Knot2     | Knot3    | etc      | (NPTUtrim<br>+ NORUtrim<br>entries)                           |                 |    |

Example:

|        |     |         |          |     |     |         |         |     |  |
|--------|-----|---------|----------|-----|-----|---------|---------|-----|--|
| GMNURB | 901 | 9       | 5        | 4   | 4   | 20      | 5       |     |  |
|        |     | 101     | 102      | 103 | 104 | 105     | 106     | 107 |  |
|        |     | 108     | 109      | 110 | 111 | 112     | 113     | 114 |  |
|        |     | 115     | 116      | 117 | 118 | 119     | 120     | 121 |  |
|        |     | 122     | 123      | 124 | 125 | 126     | 127     | 128 |  |
|        |     | 129     | 130      | 131 | 132 | 133     | 134     | 135 |  |
|        |     | 136     | 137      | 138 | 139 | 140     | 141     | 142 |  |
|        |     | 143     | 144      | 145 |     |         |         |     |  |
|        |     | 1       | 1        | 1   | 1   | 1       | 1       | 1   |  |
|        |     | 1       | 1        | 1   | 1   | 1       | 1       | 1   |  |
|        |     | 1       | 1        | 1   | 1   | 1       | 1       | 1   |  |
|        |     | 1       | 1        | 1   | 1   | 1       | 1       | 1   |  |
|        |     | 1       | 1        | 1   | 1   | 1       | 1       | 1   |  |
|        |     | 1       | 1        | 1   |     |         |         |     |  |
|        |     | 0       | 0        | 0   | 0   | 0.16667 | 0.33333 | 0.5 |  |
|        |     | 0.66667 | 0.833333 | 1   | 1   | 1       | 1       | 0   |  |
|        |     | 0       | 0        | 0   | 0.5 | 1       | 1       | 1   |  |
|        |     | 1       |          |     |     |         |         |     |  |

| Descriptor                   | Meaning   |
|------------------------------|---|
| ID                           | Identification number of a surface defined by NURBS. ID is called out on a BCBODY entry with a NURBS2 header. (Integer > 0; Required)   |
| NPTU                         | Absolute value of the number of control points. Enter NPTU as a positive number if the control points are to be input using GRID points. Enter NPTU as a negative number if the control points are to be entered using x,y,z. (Integer > 0; Required) |
| NPTV                         | Number of control points in V direction. (Integer > 0; Required)  |
| NORU                         | Order along U direction. (Integer > 0; Required)  |
| NORV                         | Order along V direction. (Integer > 0; Required)  |
| NSUBU                        | Number of subdivisions in U direction. (Integer > 0; Required)  |
| NSUBV                        | Number of subdivisions in V direction. (Integer > 0; Required)  |
| NTRIM                        | Number of trimming curves. (Integer $\geq 0$ or blank)  |
| G1, G2, G3, etc.             | Grid point IDs defining control points (Integer > 0; Required). There must be NPTU*NPTV entries.  |
| X1, Y1, Z1, X2, Y2, Z2, etc. | Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.  |

| Descriptor                     | Meaning   |
|--------------------------------|---|
| Homo1,<br>Homo2,<br>Homo3, etc | Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)                         |
| Knot1, Knot2,<br>Knot3, etc    | Knot vectors (0.0 to 1.0) in the V direction. There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)   |
| IDtrim                         | ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0) |
| NPUTtrim                       | Number of control points for this trimming vector. (Integer > 0)                                      |
| NORUtrim                       | Order for this trimming vector. (Integer > 0)   |
| NSUBtrim                       | Number of subdivisions for this trimming vector. (Integer > 0)  |
| Xisoparam                      | First coordinate of point in isoparametric space. (Real)  |
| Ysoparam                       | Second coordinate of point in isoparametric space (Real)  |
| Homo1,<br>Homo2,<br>Homo3, etc | Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)  |
| Knot1, Knot2,<br>Knot3, etc    | Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)    |

### Remarks:

1. GMNURB is recognized only in SOL 600.
2. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e. in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
3. For NURBS, enter either IDs of NPTU\*NPTV grid points as G1, G2, G3 etc. (set positive NPTU) or coordinates of abs(NPTU)\*NPTV grid point as (X1, Y1, Z1), (X2, Y2, Z2), etc. (set negative NPTU).

**GMQVOL****Define Volumetric Heat Loads**

Defines a rate of volumetric heat generation in a conduction element.

**Format:**

| 1      | 2    | 3      | 4      | 5 | 6    | 7    | 8    | 9    | 10 |
|--------|------|--------|--------|---|------|------|------|------|----|
| GMQVOL | LID  | METHOD | FIELD1 |   | EID1 | EID2 | EID3 | EID4 |    |
|        | EID5 | etc.   |        |   |      |      |      |      |    |

**Example:**

|        |     |       |    |  |    |    |   |  |  |
|--------|-----|-------|----|--|----|----|---|--|--|
| GMQVOL | 100 | TABLE | 20 |  | 18 | 23 | 7 |  |  |
|--------|-----|-------|----|--|----|----|---|--|--|

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

|        |  |                 |          |
|--------|--|-----------------|----------|
| LID    | Load set identification number.  | Integer > 0     | Required |
| METHOD | Method used to specify the data (EQUATION, TABLE, CONSTANT). See Remark 2. | Character       | Required |
| FIELD1 | Volumetric heat load data. See Remark 2.                                   | Integer or Real | Required |
| EIDi   | A list of heat conduction elements.  | Integer > 0     | Required |

**Remarks:**

- For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
- METHOD specifies the data type of FIELD1 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

| METHOD   | FIELD1  |
|----------|---|
| EQUATION | ID of a DEQATN entry defining the volumetric heat generation rate as a function of location (Integer > 0).  |
| TABLE    | ID of a TABLE3D entry defining the volumetric heat generation rate as a function of location (Integer > 0). |
| CONSTANT | Value of volumetric heat generation rate (Real).  |

Note that the fifth field will be reserved for the future development of temperature dependent functions.

**GMSPC**

## General Constraint Definition

Defines constraints for entities.

Format:

| 1     | 2   | 3 | 4      | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|--------|----|---|---|---|---|----|
| GMSPC | SID | C | ENTITY | ID |   |   |   |   |    |

Example:

|       |    |   |        |     |  |  |  |  |  |
|-------|----|---|--------|-----|--|--|--|--|--|
| GMSPC | 12 | 1 | FEEDGE | 109 |  |  |  |  |  |
|-------|----|---|--------|-----|--|--|--|--|--|

| Descriptor | Meaning  |                                |          |
|------------|--|--------------------------------|----------|
| SID        | SPC set identification number.   | Integer > 0                    | Required |
| C          | Component number in the global coordinate system.  | $0 \leq \text{Integer} \leq 6$ | 0        |
| ENTITY     | Entity that the enforced displacement is applied to<br>(Specify GRID, FEEDGE, GMCURV, FEFACE, or<br>GMSURF). See Remark 4. | Character                      | Required |
| ID         | ID of the entity selected above.   | Integer > 0                    | Required |

Remarks:

1. The components of motion specified by C (field 3) of all degrees-of-freedom associated with an entity will be constrained.
2. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set.
3. The component C has to be a single integer (1 or 2 or 3, etc.). Use multiple GMSPC entries for constraining multiple components.
4. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is the same as for the constraints. See Remark 10 under [GMBC, 2072](#) for a description of the hierarchy.
5. Coordinate transformations should not be applied to grid points on FEEDGE, GMCURVE, FEFACE or GMSURF.

**GMSURF****Surface Definition**

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

**Format:**

| 1      | 2                                  | 3     | 4     | 5     | 6 | 7 | 8 | 9 | 10 |
|--------|------------------------------------|-------|-------|-------|---|---|---|---|----|
| GMSURF | SURFID                             | GROUP | CIDIN | CIDBC |   |   |   |   |    |
|        | Evaluator Specific Data and Format |       |       |       |   |   |   |   |    |

**Example:**

|        |          |         |     |     |     |     |         |  |  |
|--------|----------|---------|-----|-----|-----|-----|---------|--|--|
| GMSURF | 101      | MSCGRP0 |     |     |     |     |         |  |  |
|        | RPC,     | POINT   |     |     |     |     |         |  |  |
|        | 0.0, 2.0 | 1.0     | 1.0 |     | 2.0 | 3.0 | 4.0,1.0 |  |  |
|        | 0.0, 2.0 | 1.0     | 1.0 | 1.0 | 2.0 | 3.0 | 4.0     |  |  |

| Descriptor | Meaning  |                  |          |  |
|------------|--|------------------|----------|--|
| SURFID     | Surface Identification number. See Remark 2.   | Integer > 0      | Required |  |
| GROUP      | Group of curves/surfaces that this surface belongs to. See Remarks 5. through 9.   | Character        | Required |  |
| CIDIN      | Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular. | Integer $\geq 0$ | 0        |  |
| CIDBC      | Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.           | Integer $\geq 0$ | 0        |  |

**Remarks:**

1. All SURFIDs must be unique.
2. A GMSURF entry is required if:
  - the geometry of the surface defined by this entry is to be used by an element.
  - output (global) coordinate system is assigned to a GMSURF.
  - permanent constraints are specified for a GMSURF.
  - loads are applied to a GMSURF.
  - enforced boundary conditions are applied to a GMSURF.
3. GMSURF is used to calculate geometric information only. The edges of the finite elements that are connected to the surface will be parametric cubic curves that are calculated from the more complex surface geometry.

4. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
5. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
6. The GROUP value is initialized by an CONNECT GEOMEVAL command in the FMS section. This command specifies the evaluator that will be used for this surface.
7. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized in the FMS Section.
8. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this surface. In this case the evaluator specific data in lines 2 through 9 of this Bulk Data entry should be provided as given on the following page. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first field.

| 1      | 2      | 3       | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|--------|---------|--------|--------|--------|--------|--------|--------|----|
| GMSURF | SURFID | MSCGRP0 | CIDIN  | CIDOUT |        |        |        |        |    |
|        | RPC    | REPRES  |        |        |        |        |        |        |    |
|        | XW(1)  | XW(2)   | XW(3)  | XW(4)  | XW(5)  | XW(6)  | XW(7)  | XW(8)  |    |
|        | XW(9)  | XW(10)  | XW(11) | XW(12) | XW(13) | XW(14) | XW(15) | XW(16) |    |
|        | YW(1)  | YW(2)   | YW(3)  | YW(4)  | YW(5)  | YW(6)  | YW(7)  | YW(8)  |    |
|        | YW(9)  | YW(10)  | YW(11) | YW(12) | YW(13) | YW(14) | YW(15) | YW(16) |    |
|        | ZW(1)  | ZW(2)   | ZW(3)  | ZW(4)  | ZW(5)  | ZW(6)  | ZW(7)  | ZW(8)  |    |
|        | ZW(9)  | ZW(10)  | ZW(11) | ZW(12) | ZW(13) | ZW(14) | ZW(15) | ZW(16) |    |
|        | W(1)   | W(2)    | W(3)   | W(4)   | W(5)   | W(6)   | W(7)   | W(8)   |    |
|        | W(9)   | W(10)   | W(11)  | W(12)  | W(13)  | W(14)  | W(15)  | W(16)  |    |

| Descriptor                 | Meaning  | Type      | Default  |
|----------------------------|--|-----------|----------|
| RPC                        | Rational Parametric Cubic Surface.                       | Character | Required |
| REPRES                     | Representation of the curve, (ALGEBRAIC, POINT, BEZIER). | Character | Required |
| XW(1)<br>through<br>XW(16) | Data used to define the surface.                         | Real      | Required |

- A rational parametric surface is defined as

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}$$

$$y(u, v) = \frac{yw(u, v)}{w(u, v)}$$

$$z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

$$0.0 \leq u \leq 1.0$$

$$0.0 \leq v \leq 1.0$$

- For REPRES = “ALGEBRAIC”, the rational parametric surface is defined by the algebraic coefficient for rational cubic equations.
- Expressed as a tensor product

$$P_{(u, v)} = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} [\text{algebraic coefficients}] \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

for the Bulk Data input in algebraic form

$$xw(u, v) = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} \begin{bmatrix} XW(1) & XW(2) & XW(3) & XW(4) \\ XW(5) & XW(6) & XW(7) & XW(8) \\ XW(9) & XW(10) & XW(11) & XW(12) \\ XW(13) & XW(14) & XW(15) & XW(16) \end{bmatrix} \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

and similarly for  $yw(u, v)$ ,  $zw(u, v)$ , and  $w(u, v)$ .

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

where  $XW(i)$ ,  $YW(i)$ ,  $ZW(i)$ , and  $W(i)$  are the algebraic coefficients for the independent equations  $xw(u, v)$ ,  $yw(u, v)$ ,  $zw(u, v)$ , and  $w(u, v)$ .

- For REPRES = “BEZIER”, the surface is defined by 16 rational Bezier control points  $V_{11}$  through  $V_{44}$ .

and

$$P(u, v) = \sum_{i=1}^4 \sum_{j=1}^4 V_{ij} B_{i,4}u \cdot B_{j,4}v$$

where  $B_{i,4}u$  and  $B_{j,4}v$  are the Bernstein polynomials for curves of degree 3.

For Bulk Data input defined as Bezier control points

$$xw(u, v) = \sum_{i=1}^4 \sum_{j=1}^4 XWi,j B_{i,4}u \cdot B_{j,4}v$$

| $V_{11}$ | $V_{12}$ | $V_{13}$ | $V_{14}$ |
|----------|----------|----------|----------|
| $V_{21}$ | $V_{22}$ | $V_{23}$ | $V_{24}$ |
| $V_{31}$ | $V_{32}$ | $V_{33}$ | $V_{34}$ |
| $V_{41}$ | $V_{42}$ | $V_{43}$ | $V_{44}$ |

where  $XW(1)$  through  $XW(16)$  are ordered to conform to the two-dimensional array for  $V_{ij}$ ; that is,  $XW(4 \cdot (i-1) + j)$  corresponds to  $xw(u, v)$  for  $V_{ij}$ . For example,  $XW(7)$  corresponds to  $V_{23}$ ,  $yw(u, v)$ ;  $zw(u, v)$ ; and  $w(u, v)$  are solved in a similar fashion.

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

- When the point representation is used, the surface is defined by 16 uniformly spaced rational points lying on the surface.

| $V_{11}$ | $V_{12}$ | $V_{13}$ | $V_{14}$ |
|----------|----------|----------|----------|
| $V_{21}$ | $V_{22}$ | $V_{23}$ | $V_{24}$ |
| $V_{31}$ | $V_{32}$ | $V_{33}$ | $V_{34}$ |
| $V_{41}$ | $V_{42}$ | $V_{43}$ | $V_{44}$ |

where  $XW(1)$  through  $XW(16)$  are ordered to contain to the two-dimensional mapping above for  $V_{ij}$ ; that is,  $XW(4 \cdot (i - 1) + j)$  corresponds to  $xw(u, v)$  for  $V_{ij}$ . For example,  $XW(7)$  corresponds to  $V_{23}$ .

- If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this surface. In this case the evaluator specific data should be on the continuation entries. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

| 1              | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9 | 10 |
|----------------|--------|--------|--------|--------|--------|--------|--------|---|----|
| EQUATION, MINU |        | MAXU   | MINV   | MAXV   | IDX    | IDY    | IDZ    |   |    |
| IDDXU          | IDDYU  | IDDZU  | IDDXV  | IDDYV  | IDDZV  | IDDXU2 | IDDYU2 |   |    |
| IDDZU2         | IDDXV2 | IDDYV2 | IDDZV2 | IDDXUV | IDDYUV | IDDZUV |        |   |    |

| Descriptor                | Meaning  | Type             | Default  |
|---------------------------|--|------------------|----------|
| EQUATION                  | EQUATION method is to be used.   | Character        |          |
| MINU, MAXU                | Range of the first parameter describing the surface. If MAXU is found less than MINU, the range for U is assumed to be $[-\infty, +\infty]$ .  | Real             | 0.0,1.0  |
| MINV, MAXV                | Range of the second parameter describing the surface. If MAXV is found less than MINV, the range is assumed to be $[-\infty, +\infty]$ .   | Real             | 0.0,1.0  |
| IDX,<br>IDY,<br>IDZ       | ID of DEQATN entries providing equations for the X,Y,Z coordinate of the surface in terms of two parameters $u$ and $v$ .  | Integer > 0      | Required |
| IDDXU, IDDYU,<br>IDDZU    | ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the first surface parameter $u$ . If a value of 0 is used, the derivatives are computed numerically. | Integer $\geq 0$ | 0        |
| IDDXV, IDDYV,<br>IDDZV    | ID of an DEQATN entry describing the first derivatives of X, Y, Z functions with respect to the first surface parameter $v$ . If a value of 0 is used, the derivatives are computed numerically.           | Integer $\geq 0$ | 0        |
| IDDXU2, IDDYU2,<br>IDDZU2 | ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the first surface parameter $u$ . If a value of 0 is used, the derivatives are computed numerically.            | Integer $\geq 0$ | 0        |

| Descriptor                | Meaning  | Type             | Default |
|---------------------------|--|------------------|---------|
| IDDXV2, IDDYV2,<br>IDDZV2 | ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the second surface parameter $v$ . If a value of 0 is used, the derivatives are computed numerically.           | Integer $\geq 0$ | 0       |
| IDDXUV, IDDYUV,<br>IDDZUV | ID of an DEQATN entry describing the mixed second derivatives of X,Y,Z functions with respect to the surface parameters $u$ , and $v$ . If a value of 0 is used, the derivatives are computed numerically. | Integer $\geq 0$ | 0       |

10. When an external geometry evaluator class is selected for this group (which is the case when the CONNECT GEOMEVAL statement selects an external geometry evaluator for the specified group), the data in Fields 1 to n will not be interpreted. In this case an image of this entry is passed on to the evaluator modules provided by the user for the specific geometric package that being used. These modules are connected with Nastran during execution. If these modules are not provided, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

- CONNECT GEOMEVAL FENDER,CATIA,’/u/kiz/data’, Version=68 as of 1/3/94
- then the GMSURF entry could use that geometry data base as follows:

| 1      | 2       | 3      | 4   | 5   | 6   | 7  | 8 | 9 | 10 |
|--------|---------|--------|-----|-----|-----|----|---|---|----|
| GMSURF | 765     | FENDER |     |     |     |    |   |   |    |
|        | Extrude | /u/kiz | 2.5 | arc | 2.7 | 66 |   |   |    |

- In this case, “Extrude u/kiz 2.5 arc 2.7 66” is passed to the geometry evaluator supplied by the user, and it is expected that the user-supplied routines interpret and use this record.

**GRAV****Acceleration or Gravity Load**

Defines acceleration vectors for gravity or other acceleration loading.

**Format:**

| 1    | 2   | 3   | 4 | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|-----|---|----|----|----|----|---|----|
| GRAV | SID | CID | A | N1 | N2 | N3 | MB |   |    |

**Example:**

|      |   |   |      |     |     |      |  |  |  |
|------|---|---|------|-----|-----|------|--|--|--|
| GRAV | 1 | 3 | 32.2 | 0.0 | 0.0 | -1.0 |  |  |  |
|------|---|---|------|-----|-----|------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| CID        | Coordinate system identification number. (Integer $\geq 0$ ; Default = 0)  |
| A          | Acceleration vector scale factor. (Real)   |
| Ni         | Acceleration vector components measured in coordinate system CID. (Real; at least one Ni $\neq 0.0$ )  |
| MB         | Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 10. (Integer; Default = 0) |

**Remarks:**

1. The acceleration vector is defined by  $\vec{a} = A\vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\vec{a}$  is equal to A times the magnitude of  $\vec{N}$ . The static loads generated by this entry are in the direction of  $\vec{a}$ .
2. A CID of zero references the basic coordinate system.
3. Acceleration or gravity loads may be combined with “simple loads” (e.g., FORCE, MOMENT) only by specification on a LOAD entry. That is, the SID on a GRAV entry may not be the same as that on a simple load entry.
4. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

5. At most nine GRAV entries can be selected in a given run either by Case Control or the LOAD Bulk Data entry. Multiples or reflections of a given acceleration or gravity load can be economically accomplished by use of the LOAD Bulk Data entry.

6. In cyclic symmetry solution sequences, the T3 axis of the coordinate system referenced in field 3 must be parallel to the axis of symmetry. In dihedral cyclic symmetry (where STYPE = "DIH" on the CYSYM entry), the T1 axis must, in addition, be parallel to Side 1 of segment 1R of the model.
7. For image superelements, the coordinate system must be rotated if the image is rotated relative to its primary superelement.
8. Acceleration or gravity loads do not include effects due to mass on scalar points.
9. The RFORCE entry may be used to specify rotational accelerations.
10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

**GRDSET****GRID Entry Defaults**

Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

**Format:**

| 1      | 2 | 3  | 4 | 5 | 6 | 7  | 8  | 9    | 10 |
|--------|---|----|---|---|---|----|----|------|----|
| GRDSET |   | CP |   |   |   | CD | PS | SEID |    |

**Example:**

|        |  |    |  |  |  |    |      |  |  |
|--------|--|----|--|--|--|----|------|--|--|
| GRDSET |  | 16 |  |  |  | 32 | 3456 |  |  |
|--------|--|----|--|--|--|----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| CP         | Identification number of coordinate system in which the location of the grid points are defined. (Integer $\geq 0$ or blank)   |
| CD         | Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors of the grid point are defined. (Integer $\geq -1$ or blank) |
| PS         | Permanent single-point constraints on the grid point. (Any combination of Integers 1 through 6 with no embedded blanks, or blank.)   |
| SEID       | Superelement identification number. (Integer $\geq 0$ or blank)  |

**Remarks:**

1. The contents of fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, and 9 are blank. If any of these fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired, one of the coordinate systems is basic, or the grid is assigned to the residual structure then the default may be overridden on the GRID entry by making one of fields 3, 7, 8, or 9 zero (rather than blank). Only one GRDSET entry may appear in the Bulk Data Section.
2. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).
3. At least one of the fields CP, CD, PS, or SEID must be specified.

**GRID****Grid Point**

Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

**Format:**

| 1    | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9    | 10 |
|------|----|----|----|----|----|----|----|------|----|
| GRID | ID | CP | X1 | X2 | X3 | CD | PS | SEID |    |

**Example:**

|      |   |   |     |      |     |  |     |  |  |
|------|---|---|-----|------|-----|--|-----|--|--|
| GRID | 2 | 3 | 1.0 | -2.0 | 3.0 |  | 316 |  |  |
|------|---|---|-----|------|-----|--|-----|--|--|

**Free Field Large Format Example:**

GRID\*, 2, , 1.0, -2.0,  
\*, 3.0, , 136

**Translates to:**

| 1     | 2   | 3 | 4 | 5 | 6   | 7 | 8    | 9 | 10 |
|-------|-----|---|---|---|-----|---|------|---|----|
| GRID* | 2   |   |   |   | 1.0 |   | -2.0 |   |    |
| *+    | 3.0 |   |   |   | 136 |   |      |   | +  |

The comma immediately following the -2.0 is required if NASTRAN IFPSTAR=NO is used. The comma is always recommended even if NASTRAN IFPSTAR=YES (Default) is used.

| Descriptor | Meaning   |
|------------|---|
| ID         | Grid point identification number. (0 < Integer < 100,000,000, see Remark 9.)  |
| CP         | Identification number of coordinate system in which the location of the grid point is defined. (Integer $\geq 0$ or blank*)   |
| X1, X2, X3 | Location of the grid point in coordinate system CP. (Real; Default = 0.0)   |
| CD         | Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors are defined at the grid point.<br>(Integer $\geq -1$ or blank, see Remark 3.)* |
| PS         | Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank*.)  |
| SEID       | Superelement identification number. (Integer $\geq 0$ ; Default = 0)  |

\*See the GRDSET entry for default options for the CP, CD, PS, and SEID fields.

**Remarks:**

1. All grid point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points.
2. The meaning of X1, X2, and X3 depends on the type of coordinate system CP as follows (see the CORDij entry descriptions):

| Type        | X1 | X2                 | X3               |
|-------------|----|--------------------|------------------|
| Rectangular | X  | Y                  | Z                |
| Cylindrical | R  | $\theta$ (degrees) | Z                |
| Spherical   | R  | $\theta$ (degrees) | $\phi$ (degrees) |

See [Grid Point and Coordinate System Definition](#) (Ch. 2) in the *MSC Nastran Reference Guide*, for a definition of coordinate system terminology.

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system. It is recommended that points on the z-axis not have their displacement directions defined with cylindrical or spherical coordinates. (See further remarks on the CORD1S, CORD2S, CORD1C, and CORD2C.)
4. The SEID field can be overridden by use of the SESET entry.
5. If CD = -1, then this defines a fluid grid point in coupled fluid-structural analysis. This type of point may only connect the CAABSF, CHACBR, CHACAB, CHEXA, CPENTA, and CTETRA elements to define fluid elements.
6. A zero (or blank if the GRDSET entry is not specified) in the CP and CD fields refers to the basic coordinate system.
7. In p-version analysis, the hierarchy set to resolve the conflicts arising in the global system input data is described under Remark 10 of the GMBC entry description.
8. CID can reference GMCORD type coordinate systems only when the GRID is connected to p-version elements.
9. For SOL 600, ID may range from 1 to  $2^{31} - 1$  (2147483647) if there are no OUTR options specified on the SOL 600 entry. If any OUTR option is specified the limit is 100000000.
10. For RC network solver in thermal analysis, the CD, PS and SEID are ignored.

**GRIDA**

Associative GRID for PAA

Defines an associative GRID point to be used in the COMBINE step of PAA.

**Format:**

| 1     | 2        | 3        | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|----------|----------|---|---|---|---|---|---|----|
| GRIDA | GID      | ORIG_GID |   |   |   |   |   |   |    |
|       | PARTNAME |          |   |   |   |   |   |   |    |

**Example:**

|       |                         |      |  |  |  |  |  |  |  |
|-------|-------------------------|------|--|--|--|--|--|--|--|
| GRIDA | 1                       | 1000 |  |  |  |  |  |  |  |
|       | Left_outboard_wing_tank |      |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| GID        | GRID ID to be used in the current run. (Integer > 0)                    |
| ORIG_GID   | GRID ID in Part being referenced. (Integer > 0)                         |
| PATNAMEi   | Name of Part ORIG_GID is in. (Character, C64, no internal blank spaces) |

**Remarks:**

1. GRIDA entries are used only in the COMBINE step of PAA processing. They are used to allow the other entries in the current bulk data to reference GRID points or SPOINTs in Parts that are being combined to create the Assembly.
2. ORIG\_GID must exist in the referenced Part. If the Part has been reduced, it must be a boundary GRID for that Part (in the A-set of that Part).
3. The GRIDA id may be treated the same as if it were defined using a GRID or SPOINT entry in the current run. The program will handle the connection to the referenced GRID/SPOINT in the Part.
4. Only dof which are in the Part A-set will be available for connection. If a GRIDA references a GRID in a Part, which has fewer than 6 dof in the Part A-set (dof available in the Part matrices), although the GRIDA will have 6 dof, only the dof which exist in the Part matrices will be connected to the Part.

**GRIDB****Axisymmetric Grid Point**

Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.

Format:

| 1     | 2  | 3 | 4 | 5   | 6 | 7  | 8  | 9   | 10 |
|-------|----|---|---|-----|---|----|----|-----|----|
| GRIDB | ID |   |   | PHI |   | CD | PS | IDF |    |

Example:

|       |    |  |  |      |  |   |     |    |  |
|-------|----|--|--|------|--|---|-----|----|--|
| GRIDB | 30 |  |  | 30.0 |  | 3 | 345 | 20 |  |
|-------|----|--|--|------|--|---|-----|----|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Grid point identification number. (0 < Integer < 1000000)   |
| PHI        | Azimuthal position of the fluid in degrees. (Real)  |
| CD         | Identification number of the coordinate system in which the displacements are defined at the grid point. (Integer $\geq 0$ or blank)            |
| PS         | Permanent single-point constraints associated with grid point. (Any combination of the Integers 1 through 6 with no embedded blanks, or blank.) |
| IDF        | Identification number of a RINGFL entry. (Integer $> 0$ )   |

Remarks:

1. GRIDB is allowed only if an AXIF entry is also present. The AXIF entry must define a fluid coordinate system.
2. All GRIDB identification numbers must be unique with respect to other scalar, structural, fluid and extra (EPOINT) points.
3. The referenced RINGFL entry must be present and be included in a boundary list (BDYLIST entry).
4. If no harmonic numbers on the AXIF entry are specified, no fluid elements are necessary.
5. The collection of all CD coordinate systems defined on all GRID and GRIDB entries is called the global coordinate system.
6. Fields 3, 4, and 6 are ignored, which facilitates the conversion of GRID entries to GRIDB entries. Note that the fields are the same except for fields 1 and 9 when a cylindrical coordinate system is used.

**GRIDF****Fluid Point**

Defines a scalar degree-of-freedom for harmonic analysis of a fluid.

**Format:**

| 1     | 2  | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|----|---|---|---|---|---|---|---|----|
| GRIDF | ID | R | Z |   |   |   |   |   |    |

**Example:**

|       |    |     |      |  |  |  |  |  |  |
|-------|----|-----|------|--|--|--|--|--|--|
| GRIDF | 23 | 2.5 | -7.3 |  |  |  |  |  |  |
|-------|----|-----|------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of axisymmetric fluid point. (0 < Integer < 1000000) |
| R          | Radial location of point in basic coordinate system. (Real > 0.0)          |
| Z          | Axial location of point in basic coordinate system. (Real)                 |

**Remarks:**

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. Grid points on slot boundaries are defined on GRIDS entries. Do not also define them on GRIDF entries.
4. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. Pressures will be plotted as displacements in the basic Z direction.
5. Load and constraint conditions are applied as if GRIDF were a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

**GRIDS****Slot Surface Point**

Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.

**Format:**

| 1     | 2  | 3 | 4 | 5 | 6   | 7 | 8 | 9 | 10 |
|-------|----|---|---|---|-----|---|---|---|----|
| GRIDS | ID | R | Z | W | IDF |   |   |   |    |

**Example:**

|       |    |     |      |     |  |  |  |  |  |
|-------|----|-----|------|-----|--|--|--|--|--|
| GRIDS | 25 | 2.5 | -7.3 | 0.5 |  |  |  |  |  |
|-------|----|-----|------|-----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number of the slot point. (Integer > 0)                |
| R          | Radial location of point in basic coordinate system. (Real ≠ 0.0)     |
| Z          | Axial location of point in basic coordinate system. (Real)            |
| W          | Slot width or thickness at the GRIDS point. (Real ≥ 0.0 or blank)     |
| IDF        | Identification number to define a GRIDF point. (Integer > 0 or blank) |

**Remarks:**

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID (and IDF if present) must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. If W is blank, the default value on the AXSLOT entry will be used.
4. The IDF number is referenced on the CAXIFI entry for central cavity fluid elements next to the interface. The IDF number is entered only if the grid point is on an interface. In this case, the IDF should also be defined on a GRIDF entry.
5. If IDF is nonzero, then R must be greater than zero.
6. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. The slot width, W, corresponds to the basic Z coordinate. The pressure will be plotted in the basic Z direction.
7. Load and constraint conditions are applied as if the GRIDS is a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

**GUST****Aerodynamic Gust Load Description**

Defines a stationary vertical gust for use in aeroelastic response analysis.

**Format:**

| 1    | 2   | 3     | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|------|-----|-------|----|----|---|---|---|---|----|
| GUST | SID | DLOAD | WG | X0 | V |   |   |   |    |

**Example:**

|      |     |    |     |    |      |  |  |  |  |
|------|-----|----|-----|----|------|--|--|--|--|
| GUST | 133 | 61 | 1.0 | 0. | 1.+4 |  |  |  |  |
|------|-----|----|-----|----|------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Gust set identification number. (Integer > 0)  |
| DLOAD      | Set identification number of a TLOADi or RLOADi entry that defines the time or frequency dependence. (Integer > 0) |
| WG         | Scale factor (gust velocity/forward velocity) for gust velocity. (Real ≠ 0.0)                                      |
| X0         | Streamwise location in the aerodynamic coordinate system of the gust reference point. (Real)                       |
| V          | Velocity of vehicle. See Remark 5. (Real > 0.0)  |

**Remarks:**

1. The GUST entry must be selected with the Case Control command GUST = SID.
2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is

$$WG \cdot T\left(t - \frac{X-X0}{V}\right)$$

where  $T$  is the tabular function.

3. In random analysis, a unit gust velocity (WG = 1/velocity) is suggested. The actual rms value is entered on the TABRNDG entry.
4. X0 and V may not change between subcases under one execution.
5. V must be equal to VELOCITY on the AERO Bulk Data entry.

**HADACRI****Mesh Adaptivity Criterion and Corresponding Parameters - SOL 101 & 400 (Linear)**

Specifies Mesh refinement criterion for adaptive mesh refinement and corresponding parameters.

**Format:**

| 1       | 2      | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|--------|------|----|----|----|----|----|----|----|
| HADACRI | CRITID | TYPE | F1 | F2 | F3 | F4 | F5 | F6 |    |

**Example:**

|         |   |   |     |  |  |  |  |  |  |
|---------|---|---|-----|--|--|--|--|--|--|
| HADACRI | 1 | 1 | 0.9 |  |  |  |  |  |  |
|---------|---|---|-----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| CRITID     | Identification number referenced by the CRITID field in the HADAPTL Bulk Data entry. (Integer $> 0$ ; no Default) |
| TYPE       | Type of Mesh refinement adaptivity criteria. See Remark 1. (Integer $\geq 1$ and $\leq 4$ ; Default = 1)          |
| F1 to F8   | Criteria specific parameters. See Remark 2. (Real; no Default)  |

**Remarks:**

1. The mesh refinement criteria currently available (and selected in the TYPE field) are:

| TYPE | Name of Mesh Refinement Criterion         |
|------|---|
| 1    | Error indicator based                     |
| 2    | Element within a spatial spherical region |
| 3    | Elements within a spatial cubic region    |
| 4    | Elements in contact criteria              |

2. The following table describes the different refinement criteria and corresponding parameters:

| TYPE | Description  |
|------|--|
| 1    | <p>In this case a scalar error indicator <math>E_e</math> is computed for all elements '<math>e</math>' in the finite element mesh. Then, an element '<math>e</math>' will be refined if</p> $E_e^2 \geq F_1 \bar{E}^2$ <p>where <math>F_1</math> is a weight factor (<math>0 \leq F_1 \leq 1</math>) specified in the F1 field and <math>\bar{E}</math> is the quadratic mean of the error indicator defined as</p> $\bar{E}^2 = \frac{1}{N} \sum_{e=1}^N E_e^2$ <p>with <math>N</math> the total number of elements in the element set where element '<math>e</math>' belongs. For this criteria the fields F2 to F6 are ignored.</p>  |
|      | <p>The elemental error indicator <math>E_e</math> is computed using the grid point stresses following the procedure utilized by the ELSDCON Case Control command and described in <a href="#">Mesh Stress Discontinuities at Grid Points</a> in the <i>MSC Nastran Reference Guide</i>. This procedure can be summarized as follows:</p> <ul style="list-style-type: none"> <li>■ Let</li> <math display="block">\sigma_{aij} = \sum_{e=1}^{N_a} W_a^e \sigma_{aij}^e</math> <p>be the weighted average over all elements '<math>e</math>' concurrent to a given node '<math>a</math>' of each component '<math>ij</math>' of the grid point stresses <math>\sigma_{aij}^e</math> where <math>W^e</math> is a weighting factor assigned to element '<math>e</math>' and <math>N_a</math> is the number of elements connected to the given node '<math>a</math>'.</p> <li>■ An estimate of the error in a particular component of stress '<math>ij</math>' at a grid point '<math>a</math>' is then be computed as</li> <math display="block">E_{aij}^2 = \sum_{f=1}^N W_a (\sigma_{aij}^e - \sigma_{aij})^2</math> <li>■ Averaging the latter over the different stress components, '<math>ij</math>', over the different shell fibers (for shell elements) and over the different grid points '<math>a</math>' connected by a given element '<math>e</math>' we obtain the elemental, scalar error indicator <math>E_e</math>.</li> </ul> |

| TYPE | Description   |
|------|---|
| 2    | In this case the user specifies a spherical region in space with center given by (F1,F2,F3) and radius given by F4. Then, all elements with at least one node with basic coordinates (X,Y,Z) within the spherical region (i.e., such that $\ (X, Y, Z) - (F1, F2, F3)\  < F4$ ) will be refined. For this criteria the fields F5 and F6 are ignored.                    |
| 3    | In this case the user specifies a hexahedral region in space, aligned with the basic coordinates system, with corners given by (F1,F2,F3) and (F4,F5,F6). Then, all elements with at least one node with basic coordinates (X,Y,Z) within the specified hexahedral region (i.e., such that $F1 \leq X \leq F4, F2 \leq Y \leq F5, F3 \leq Z \leq F6$ ) will be refined. |
| 4    | In this case all elements with at least one node involved in contact either as touching or touched nodes in deformable contact bodies are refined. For this criteria the fields F1 to F6 are ignored.   |

3. Each criteria must have a unique ID (specified by the CRITID field and referenced by the CRITID field of the Bulk Data entry, [HADAPTL, 2126](#)).
4. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI, 1005](#)).

**HADAPTL**

## Local Adaptive Mesh Refinement Control Parameters - SOL 101 &amp; 400 (Linear)

Specifies Local Adaptive Mesh Refinement control parameters.

**Format:**

| 1       | 2        | 3 | 4 | 5      | 6      | 7            | 8       | 9        | 10 |
|---------|----------|---|---|--------|--------|--------------|---------|----------|----|
| HADAPTL | ID       |   |   | REPEAT | CRITID | WHEREME<br>T | WHEREID | SNAPMETH |    |
|         | MAXLEVEL |   |   |        |        |              |         |          |    |

**Example:**

|         |   |  |  |    |   |      |   |  |  |
|---------|---|--|--|----|---|------|---|--|--|
| HADAPTL | 1 |  |  | 10 | 1 | PROP | 5 |  |  |
|---------|---|--|--|----|---|------|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number referenced by the Case Control HADAPT command. (Integer > 0; no Default)   |
| REPEAT     | Maximum number of analysis performed before the adaptive mesh refinement process is stopped. See Remark 3. (Integer $\geq 0$ ; Default = 6)  |
| CRITID     | Identification number of a mesh refinement criterion to be selected with the HADACRI Bulk Data entry. See Remarks 1. and 2. and the HADACRI Bulk Data entry. (Integer > 0; no Default)   |
| WHEREMET   | Method to specify the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. It can take the values: "ALL" or "SUPER" or "PROP". See Remark 4. (Character; Default = ALL)  |
| WHEREID    | ID of the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. Must be specified if WHEREMET is "SUPER" or "PROP". If WHEREMET=ALL, this field will be ignored. See Remark 4. (Integer $\geq 0$ ; Default = 0) |
| SNAPMETH   | Method to project, snap or relax new grid points created on mid-edge or mid-faces on the mesh boundary during the refinement process onto the analysis domain boundary:  |
| 0          | No projection; New grids are placed in the mid-side of edges.  |
| 1          | New grid points are projected onto a smooth approximation of the analysis domain boundary interpolated from the initial mesh boundary.<br>(Integer $\geq 0$ ; Default = 0)   |
| MAXLEVEL   | Maximum refinement level allowed for each individual element in the mesh. No elements in the mesh will be refined to a level bigger than MAXLEVEL. (Integer > 0; Default = REPEAT)   |

**Remarks:**

1. The adaptive mesh refinement occurs when a particular refinement criterion is satisfied. Data for the refinement criterion is specified by the Bulk Data entry HADACRI referenced by the CRITID field.

2. Multiple mesh refinement criteria can be selected in different subsets of the model. To this end, the user needs to define multiple HADAPTL entries with the same ID. Each entry might specify a different criteria (referenced in the CRITID field and defined on the corresponding HADACRI Bulk Data entry) on different subsets of the mesh (defined in the WHEREMET and WHEREID fields).
3. When multiple HADAPTL entries with the same ID are specified, NASTRAN will chose for the REPEAT, SNAPMETH, and MAXLEVEL field the maximum among all multiple instances.
4. The fields WHEREMET and WHEREID refer respectively to the Method to specify the mesh refinement region (subjected to the refinement criterion referenced by the field CRITID and defined with the Bulk Data entry, [HADACRI, 2123](#)) and its corresponding ID. For example, WHEREMET=SUPER, WHEREID=3 means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in superelement 3. Likewise, WHEREMET=PROP, WHEREID=5 (see the previous Example) means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in those element with Property ID equal to 5. Finally, WHEREMET=ALL imply mesh refinement in all elements.
5. In partitioned superelements, the HADAPT entry must be specified in the main bulk data section. Entries specified in the Bulk Data Section corresponding to individual parts (sections beginning with BEGIN SUPER) will be ignored.
6. When using regular superelements, the Bulk Data Section must begin with BEGIN SUPER as opposed to BEGIN BULK, in order for the refinement to be appropriately propagated across superelement boundaries. If BEGIN BULK is used, grid points on the superelement boundaries will be duplicated and not shared by the joining superelements.
7. The user should avoid the use of MPC sets 90000000 to 99999999 which are reserved for hanging nodes constraints generated during the adaptive mesh refinement process.
8. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI, 1005](#)).
9. When SNAPMETH=0, all mid-edge nodes belonging to straight edges are placed on the mid-side of its edge. By contrast, when SNAPMETH=1, mid-edge nodes belonging to the boundary of the mesh are projected to a smooth approximation of the analysis domain boundary interpolated from the mesh boundary.
10. Mid-face nodes belonging to bilinear quadrilateral faces are placed at the baricenter of its face.

**HEATLOS**

## Heat Loss Through Convection or Radiation of the Airbag Surface - SOL 700

Defines the heat-transfer model to be used with GBAG or COUPLE. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5       | 6        | 7     | 8      | 9 | 10 |
|---------|-----|-------|-------|---------|----------|-------|--------|---|----|
| HEATLOS | HID | HTRID | SUBID | HTRTYPE | HTRTYPID | COEFF | COEFFV |   |    |

**Example:**

|         |     |    |  |         |   |       |    |  |  |
|---------|-----|----|--|---------|---|-------|----|--|--|
| HEATLOS | 101 | 83 |  | HTRCONV | 2 | TABLE | 14 |  |  |
|---------|-----|----|--|---------|---|-------|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| HID        | Unique number of a HEATLOS entry. (Integer > 0; Required)  |
| HTRID      | Number of a set of HEATLOS entries HTRID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)   |
| SUBID      | (Integer $\geq 0, 0$ ) <ul style="list-style-type: none"> <li>&gt; 0 Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface as defined on the GBAG or COUPLE entries.</li> <li>= 0 HEATLOS definitions are used for the entire surface as defined on the GBAG or COUPLE entries.</li> </ul>  |
| HTRTYPE    | Defines the type of heat transfer. (Character; Required)   |
| HTRCONV    | The HTRCONV logic is used to model heat transfer through convection in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface. |
| HTRRAD     | The HTRRAD logic is used to model heat transfer through radiation in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface.   |
| HTRTYPID   | Heat transfer ID. References existing HTRTYPE entry. (Integer > 0; Required)   |
| COEFF      | Method of defining the area coefficient. (Character, CONSTANT)   |
| CONSTANT   | The area coefficient is constant and specified on COEFFV.  |

| Descriptor | Meaning   |
|------------|---|
| TABLE      | The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.                        |
| COEFFV     | The area coefficient or the number of a TABLED1 entry depending on the COEFF entry.<br>$(0.0 \leq R \leq 1.0 \text{ or } 1 > 0, 1.0)$ |

### Remarks

1. A combination of multiple HEATLOS with different HTRTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to HEATLOS) or an Eulerian model (COUPLE to HEATLOS). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).
3. For the same BSURF multiple, different types of heat transfer may be defined.
4. A more detailed description can be found in Porosity in Air Bag for more details.

**HGSUPPR**

## Hourglass Suppression Method - SOL700

Defines the hourglass suppression method and the corresponding hourglass damping coefficients. Used in SOL 700 only.

**Format:**

| 1       | 2      | 3     | 4   | 5     | 6      | 7      | 8      | 9      | 10 |
|---------|--------|-------|-----|-------|--------|--------|--------|--------|----|
| HGSUPPR | HID    | PROP  | PID | HGTYP | HGCMEM | HGCWRP | HGCTWS | HGCSOL |    |
| +       | RBRCOR | VALUE |     |       |        |        |        |        |    |

**Example:**

|         |     |       |     |     |     |     |     |  |  |
|---------|-----|-------|-----|-----|-----|-----|-----|--|--|
| HGSUPPR | 1   | SHELL | 100 | FBV | 0.1 | 0.1 | 0.1 |  |  |
| +       | YES | 10000 |     |     |     |     |     |  |  |

| Descriptor | Meaning   |   |
|------------|---|---|
| HID        | Hourglass suppression definition number. (Integer > 0; Required.) |   |
| PROP       | Property type. (Character; See Remark 1.)                         |   |
| PID        | Property number. (Integer > 0; See Remark 1.)                     |   |
| HGTYP      | Hourglass suppression method: (Character; See Remark 2.)          |   |
|            | FBV   | For shells only (default for shells)  |
|            | FBS   | For shells and solids (default for solids)                                  |
|            | Dyna  | For solids only   |
| HGCMEM     | Membrane damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)    |   |
| HGCWRP     | Warping damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)     |   |
| HGCTWS     | Twisting damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)    |   |
| HGCSOL     | Solid damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)       |   |
| RBRCOR     | Rigid body rotation correction: (Character; default=NO)           |   |
|            | NO  | No rigid-body rotation correction is applied to hourglass resisting forces. |
|            | YES   | Rigid-body rotation correction is applied to hourglass resisting forces.    |
| VALUE      | Number of steps. (Integer > 0; See Remark 3.)                     |   |

**Remarks:**

1. The property type definition and the property number are required. Since property numbers are unique within a certain class of element types, the property type and the property number uniquely define to what elements the hourglass suppression method and coefficients apply. The following property types are valid entries:

|       |                               |
|-------|-------------------------------|
| BAR   | For bar elements              |
| BEAM  | For beam elements             |
| BELT  | For belt elements             |
| COMP  | For composite shell elements  |
| DAMP  | For damper elements           |
| ELAS  | For spring elements           |
| EULER | For Eulerian elements         |
| ROD   | For rod elements              |
| SHELL | For shell elements            |
| SOLID | For solid Lagrangian elements |

It must be noted however, that only shell CQUAD4 and Lagrangian CHEXA and CPENTA elements can suffer from undesired hourglass modes. All HGSUPPR entries referring to other types of elements are ignored.

2. There are three types of hourglass suppression methods available in Nastran. These are standard DYNA viscous (DYNA) hourglass damping, the Flanagan-Belytschko Stiffness (FBS) hourglass damping, and the Flanagan-Belytschko Viscous (FBV) hourglass damping.  
Lagrangian solid elements can address DYNA and FBS suppression; shell elements can address DYNA and FBV suppression. The default for the Lagrangian solid elements is FBS. The default for the shell elements is FBV.
3. The rigid-body rotation correction on the hourglass forces is only necessary in cases where shell elements undergo a large rigid-body rotation. If the RBRCOR field is set to YES, and the VALUE field is left blank, the correction is applied during each time step. If the VALUE field is set to a number, the rotation correction is applied only when the rigid-body rotation would result in a rotation of the element over 90° in less than VALUE time steps. Usually, if the rigid-body rotation correction is necessary; 10000 is a good value. This option saves some CPU time.  
The RBRCOR option applies to the Key-Hoff shell formulation only; for all other element types and formulations, the option is ignored.
4. The membrane, warping and twisting coefficients apply to shell elements only; for all other element types, the data is ignored. The solid damping coefficient applies to solid Lagrangian elements only; for all other element types, the data is ignored.

**HTRCONV****Air Bag Convection**

Defines the heat transfer through convection for a COUPLE and/or GBAG surface.

Convection is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

Format:

| 1       | 2     | 3      | 4      | 5    | 6 | 7 | 8 | 9 | 10 |
|---------|-------|--------|--------|------|---|---|---|---|----|
| HTRCONV | HTRID | HTRCFC | HTRCFT | TENV |   |   |   |   |    |

Example:

|         |   |  |    |       |  |  |  |  |  |
|---------|---|--|----|-------|--|--|--|--|--|
| HTRCONV | 8 |  | 14 | 293.0 |  |  |  |  |  |
|---------|---|--|----|-------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| HTRID      | Unique number of a HTRCONV entry. Referenced from HEATLOS. (Integer > 0; Required)  |
| HTRCFC     | Constant heat transfer convection coefficient. See Remark 3. (Real $\geq 0$ )   |
| HTRCFT     | The heat transfer convection coefficient is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer $\geq 0$ ) |
| TENV       | Environmental temperature. (Real > 0; Required)   |

Remarks:

1. The HTRCONV entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gas constants ( $\gamma$ ,  $R$ ,  $c_v$  and/or  $c_p$ ) have to be defined on the EOSGAM entry.
3. Either HTRCF-C or HTRCF-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

**HTRRAD****Air Bag Radiation**

Defines the heat transfer through radiation for a COUPLE and/or GBAG surface.

Radiation is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

**Format:**

| 1      | 2     | 3        | 4        | 5    | 6      | 7 | 8 | 9 | 10 |
|--------|-------|----------|----------|------|--------|---|---|---|----|
| HTRRAD | HTRID | GASBMI-C | GASEMI-T | TENV | SBOLTZ |   |   |   |    |

**Example:**

|        |   |      |  |       |          |  |  |  |  |
|--------|---|------|--|-------|----------|--|--|--|--|
| HTRRAD | 2 | 0.15 |  | 293.0 | 5.676E-8 |  |  |  |  |
|--------|---|------|--|-------|----------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| HTRID      | Unique number of a HTRRAD entry. Referenced from HEATLOS. (Integer > 0; Required)   |
| GASEMI-C   | Constant gas emissivity. See Remark 3. (Real $\geq 0$ )   |
| GASEMI-T   | The gas emissivity is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer $\geq 0$ ) |
| TENV       | Environmental temperature. (Real > 0; Required)   |
| SBOLTZ     | Stephan-Boltzman constant. (Real > 0; Required)   |

**Remarks:**

1. The HTRRAD entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gasconstants ( $\gamma$ ,  $R$ ,  $c_v$  and/or  $c_p$ ) have to be defined on the EOSGAM entry.
3. Either GASEMI-C or GASEMI-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

**HYBDAMP**

## Hybrid Modal Damping for Direct Dynamic Solutions

Specifies hybrid damping parameters.

**Format:**

| 1       | 2  | 3      | 4     | 5     | 6      | 7 | 8 | 9 | 10 |
|---------|----|--------|-------|-------|--------|---|---|---|----|
| HYBDAMP | ID | METHOD | SDAMP | KDAMP | PRTEIG |   |   |   |    |

**Example:**

|         |     |      |      |    |     |  |  |  |  |
|---------|-----|------|------|----|-----|--|--|--|--|
| HYBDAMP | 101 | 2000 | 2001 | NO | YES |  |  |  |  |
|---------|-----|------|------|----|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number of HYBDAMP entry (Integer > 0; Required)                                      |
| METHOD     | Identification number of METHOD entry for modes calculation. (Integer > 0; Required)                |
| SDAMP      | Identification number of TABDMP1 entry for modal damping specification. (Integer > 0; Required)     |
| KDAMP      | Selects modal “structural” damping. See Remark 1. (Character: “Yes” or “NO”; Default = “NO”)        |
| PRTEIG     | Print eigenvalue summary from hybrid damping calculation (Character: “Yes” or “NO”; Default = “NO”) |

**Remarks:**

- For KDAMP = “YES”, the viscous modal damping is entered into the complex stiffness matrix as structural damping.
- Hybrid damping is generated using modal damping specified by the user on TABDMP entries.

$$BH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} b(\omega_1) & & & \\ & b(\omega_2) & & \\ & & \ddots & \\ & & & b(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

For KDAMP = “YES”

$$KH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} g(\omega_1) & & & \\ & g(\omega_2) & & \\ & & \ddots & \\ & & & g(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

where

$\phi_i$  = modes of the structure

$[M]$  = structural mass matrix

$b(\omega_i)$  = modal damping values,  $b(\omega_i) = g(\omega_i)\omega_i m_i$

$g(\omega_i)$  = twice the critical damping ratio determined from user specified TABDMP entry

$\omega_i$  = natural frequency of mode  $\phi_i$

$m_i$  = generalized mass of mode  $\phi_i$

3. Hybrid damping is based on a modal solution and the mass matrix. Even if user requested all the modes and specified a flat value of G the result is still dependent on the Rank of the Mass matrix. Also note that Hybrid damping does not include residual vectors to compensate for high frequency contribution and other possible massless degrees of freedom effects in the Rank of the mass matrix. PARAM, COUPMASS, 1 may improve the mass rank.
4. HYBDAMP is referenced by the DAMPING and ROTHYBD Bulk Data entries.
5. When hybrid damping is used during FRF component generation, it is recommended to set the KDAMP field to YES on the HYBDAMP entry. See remarks under SDAMPING Case Control description for further details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**HYDROC**

Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using fluid cavities.

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using fluid cavity data. The HYDROC entry is NOT supported in SOL101 because SOL 101 does not support the Fluid Structure Interface.

**Format:**

| 1      | 2       | 3      | 4     | 5 | 6   | 7    | 8    | 9 | 10 |
|--------|---------|--------|-------|---|-----|------|------|---|----|
| HYDROC | SID     | PSOLID | PNOM  | G | PCH | PLD4 | OVRD |   |    |
|        | CID/GRD | HGHTB  | HABOV |   |     |      |      |   |    |

**Example:**

|        |    |    |     |      |  |   |  |  |  |
|--------|----|----|-----|------|--|---|--|--|--|
| HYDROC | 66 | 33 | 42. | 980. |  | 1 |  |  |  |
|        | 99 | 0. | 10. |      |  |   |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| PSOLID     | PSOLID ID of the cavity. Fluid density will be obtained from the associated MAT10 entry. (Integer > 0); HYDROC requires ACMODL,DIFF (the default value).  |
| PNOM       | Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; default = 0.0)   |
| G          | Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)  |
| PCH        | 0 - default (no punch);<br>1 - Punch Set of structural grids.<br>2 - Punch Set of structural elements.<br>3 - Punch both structural grids and structural elements. (Integer $\geq 0$ )  |
| PLD4       | 0 - default (no punch);<br>1 - Punch the resulting PLOAD4 entries. (Integer $\geq 0$ )  |
| OVRD       | 0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued.<br>1 - Allow non unit value of G.   |
| CID/GRD    | Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer $\geq 0$ ; a coordinate system ID is used. If Integer $< 0$ ; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal error. See Remark 1.; (Integer - default 0 -basic). |

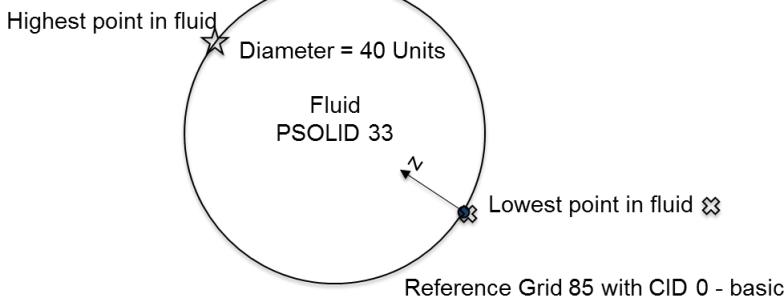
| Descriptor | Meaning   |
|------------|---|
| HGHTB      | The z-distance to the lowest fluid point measured from reference origin. See Remark 1.. (Real; Default = 0.0)                         |
| HABOV      | The height of the fluid above the lowest point measured in the z-direction from reference origin. See Remark 1. (Real; Default = 0.0) |

### Remarks:

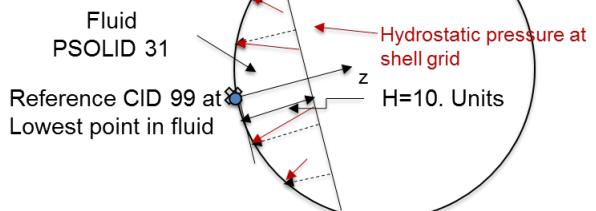
1. On the HYDROC entry, HGHTB along with the z-coordinate direction, are used to determine the location of the “bottom” of the fluid body in the case where the fluid cavity is full and there is no free surface.

In the case where a free fluid surface has been modeled in the cavity, CID/GID and HGHTB are for reference only. The lowest fluid level is determined as the lowest wetted grid in the FSI coupling measure in the direction defined by CID/GID. From the FSI coupling, free surface formed by examining wetted grids may not be perpendicular to the z-direction of CID/GID. Hence, HABOV will be considered as average free surface and hydrostatic pressure will be computed with respect to HABOV. However, a fatal message will occur if the average free surface specified by HABOV is significantly above the free surface of wetted grids. Also note that some wetted grids/elements may not get loaded if HABOV is too low.

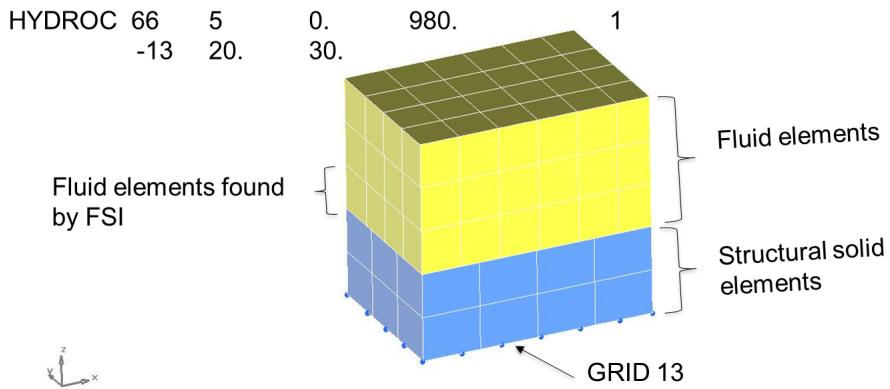
HYDROC, 95, 33, 0., 1., 0, 1, , ,  
, -85, 0., 40.



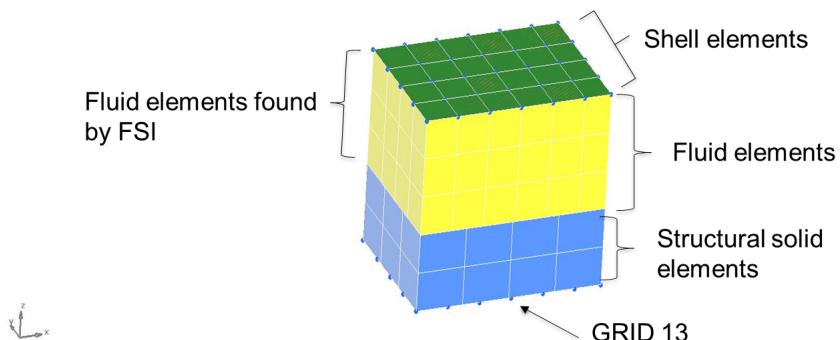
HYDROC, 95, 31, 0., 1., 0, 1, , ,  
, 99, 0., 10.



2. The HYDROC entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID
3. Each fluid cavity should have a unique PSOLID entry as HYDROC uses the PSOLID to identify the cavity.
4. The PNOM is applied only from fluid surface to bottom of fluid.
5. HYDROC interaction with the Fluid Structure Interface (FSI) algorithm is shown in the two figures below. The top figure shows a column of fluid over some structural elements. The column of fluid is 30 units high, however, only its bottom surface is in contact with structural elements. Therefore, the FSI will only see a fluid that is 10 units high. A HYDROC entry for this model, with a HABOV = 50 units would fail to compute any hydrostatic loading, because 50 units is well outside the tolerance of the FSI interface, which is at 30 units.



The second figure shows the same fluid column, but a layer of shell elements has been laid on top of the fluid column.



HYDROC 66 5 0. 980. 1  
-13 20. 50.

In this case, the FSI sees a column of fluid 50 units high. Thus a HABOV = 50 units will generate PLOAD4 entries using a height of 30 units for the fluid. For this model, any HABOV with values ( $30 \text{ units} \leq \text{HABOV} \leq 50 \text{ units}$ ) will yield valid PLOAD4 entries on the fluid structure interface of the structural solid elements with the height used depending on the value of HABOV.

6. The figures below show additional HYDROC PNOM rules associated with the FSI interface. Also note how fluid cavities are defined by PSOLID ID.

HYDROS (1) applies a hydrostatic loading of  $55. + 0.*h*0.$  interior to the cylinder with POINT 17  
 HYDROC (2) applies a hydrostatic loading  $0. + \rho * h * 980.$

HYDROC (3) applies a hydrostatic loading  $42. + \rho * h * 980.$  – above the fluid there is no pressure of 42. units

1. HYDROS, 66, 55., 0., 0., 6, ,1 , ,

, -85, 0., 40., 17

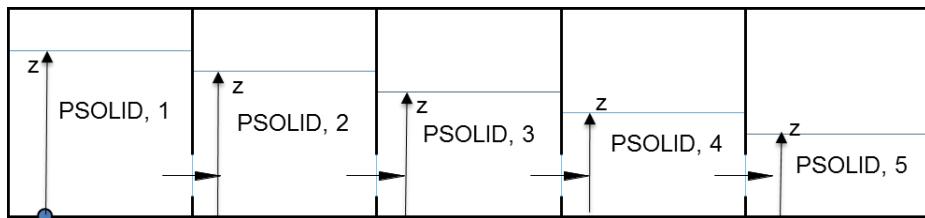
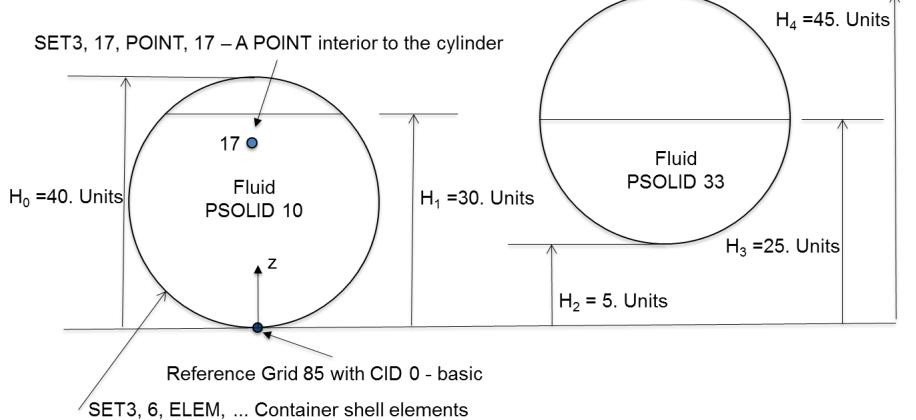
2. HYDROC, 66, 10, 0., 980., 0, 1, , ,

, -85, 0., 30.

3. HYDROC, 66, 33, 42., 980., 0, 1, , ,

, -85, 5., 25.

SET3, 17, POINT, 17 – A POINT interior to the cylinder



Reference origin with CID

Staggered grid for flooding simulation.

**HYDROS**

Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using element or grid sets

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using element set data.

**Format:**

| 1      | 2       | 3     | 4     | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|---------|-------|-------|------|------|------|------|------|----|
| HYDROS | SID     | PNOM  | RHO   | G    | SETE | SETG | PLD4 | OVRD |    |
|        | CID/GRD | HGHTB | HABOV | CNTL |      |      |      |      |    |

**Example:**

|        |     |     |      |     |     |  |   |  |  |
|--------|-----|-----|------|-----|-----|--|---|--|--|
| HYDROS | 77  | 55. | 0.08 | 17. | 300 |  | 1 |  |  |
|        | 200 |     | 97.  | 83  |     |  |   |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| PNOM       | Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; Default = 0.0)   |
| RHO        | Mass density. (Real > 0.0)  |
| G          | Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)  |
| SETE       | The ID of a SET3 entry using (DES= ELEM) or SET1 to identify the element IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer $\geq 0$ )   |
| SETG       | The ID of a SET3 entry using (DES= GRID) or SET1 to identify the element grid IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer $\geq 0$ )  |
| PLD4       | 0 - default (no punch);<br>1 - Punch the resulting PLOAD4 entries. (Integer $\geq 0$ )  |
| OVRD       | 0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued.<br>1 - Allow non unit value of G.   |
| CID/GRD    | Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer $\geq 0$ ; a coordinate system is used. If Integer $< 0$ ; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal. See Remark 1.; (Integer - default 0 -basic).<br>Note: A spherical coordinate system will result in a fatal error. |
| HGHTB      | The z-distance to the lowest fluid point measured from reference origin. (Real; Default = 0.0)  |

| Descriptor | Meaning  |
|------------|--|
| HABOV      | The height of the fluid above the lowest point measured in the z-direction from reference origin. (Real; Default = 0.0)                  |
| CNTL       | SET3 ID of control grids to select wetted surface(s). For an immersed physical body, at least one grid per quadrant. (Integer $\geq 0$ ) |

### Remarks:

1. The HYDROS entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID.

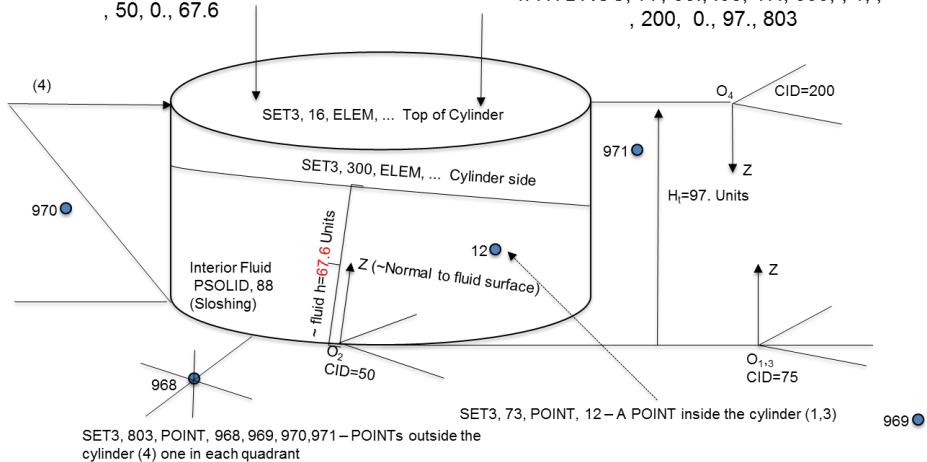
HYDROS (1) applies a hydrostatic loading of  $25. + .08*h*0.$  interior to the cylinder with POINT 12  
 HYDROC (2) applies a hydrostatic loading  $0. + p^* h * 1.$ ;  $h$  – measured normal to sloshed surface  
 HYDROS (3) applies a hydrostatic loading  $55. + .08* h * 980.$  ( $h$  is constant 97. units for this set)  
 HYDROS (4) applies a hydrostatic loading  $55. + .08*h * 17.$  for all elements on exterior cylinder side.

1. HYDROS, 77, 25., .08, 0, 300, , 1, , , 75, 0, 97., 73

3. HYDROS, 77, 55., .08, 980., 16, , 1, , , 75, 0, 97., 73

2. HYDROC, 77, 88, 0, 1., 0, 1, , , 50, 0., 67.6

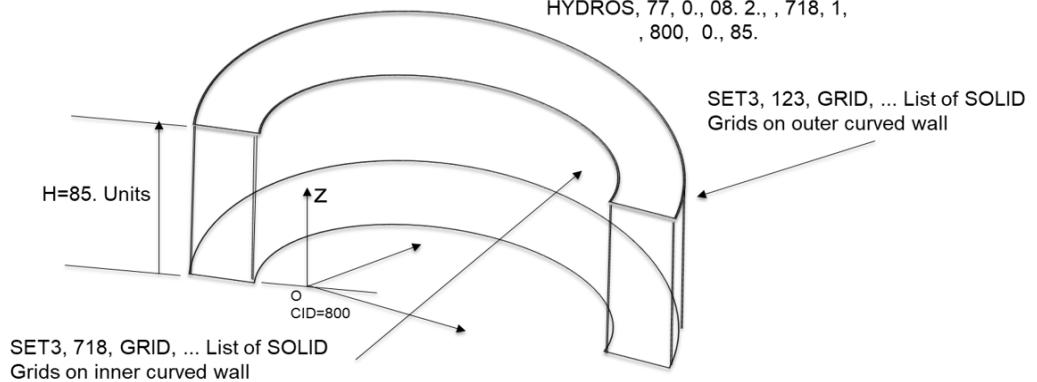
4. HYDROS, 77, 55., .08, 17., 300, , 1, , , 200, 0, 97., 803



**SOLID Model**

HYDROS, 77, 0., 08. 2., , 123, 1,  
, 800, 0., 85.

HYDROS, 77, 0., 08. 2., , 718, 1,  
, 800, 0., 85.



**HYDSTAT****Hydrostatic Preset of Density in Euler Elements**

Initializes the Euler element densities in accordance to a hydrostatic pressure profile. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9    | 10 |
|---------|-----|-----|-----|-----|-----|-----|-----|------|----|
| HYDSTAT | HID | MID | GID | CID | XCG | YCG | ZCG | PATM |    |

**Example:**

|         |     |   |  |  |     |     |     |         |  |
|---------|-----|---|--|--|-----|-----|-----|---------|--|
| HYDSTAT | 101 | 4 |  |  | 0.0 | 0.0 | 0.0 | 100000. |  |
|---------|-----|---|--|--|-----|-----|-----|---------|--|

| Descriptor      | Meaning  |
|-----------------|--|
| HID             | Identification number of the HYDSTAT entry referenced from the COUPLE entry.<br>(Integer > 0; Required)    |
| MID             | Material MATDEUL id to which the hydrodynamic pressure profile will be applied.<br>(Integer > 0; Required) |
| GID             | Number of a grid point at the free surface. See Remark 4. (Integer $\geq 0$ )                              |
| CID             | Local coordinate system. See Remark 4. (Integer $\geq 0$ )   |
| XCG,YCG,<br>ZCG | Coordinates of a point at the free surface. See Remark 4. (Real)   |
| PATM            | Pressure at free surface. (Real; Required)   |

**Remarks:**

1. It is assumed that each Euler domain contains at most two Eulerian materials and includes the GRAV entry. One material has to be a fluid using EOSPOL the other a gas or void. This EOSPOL material is given by the MID entry. The interface between gas and fluid is the free surface and is assumed to be normal to the gravity vector as specified on the GRAV entry. For example if the gravity vector points in the z-direction then the interface between the gas and the fluid has to be horizontal.
2. The hydrostatic preset changes the density of the fluid like material in order to conform to the hydrostatic preset. It overrules the material densities as specified on the TICEL and TICVAL entries. Densities of the gas like material are not changed.
3. The free surface has to match with material fractions as defined in the initialization of Euler elements by the TICEL and TICEUL entries. The hydrostatic preset only changes densities, it does not change material fractions.
4. There are two options to enter the location of the free surface. The first option is to enter a grid point number. In that case the fields CID and XCG-ZCG have to be left blank. The GRIDPOINT entry already has the option of using a local coordinate system. When coordinates are used then the field GID has to be left blank.

5. If there is no structural grid point indicating the free surface then a new grid point can be defined that will only be used for determining the free surface level. If the Gravity vector points in the z-direction, only the z-coordinate of the grid point will be used. The x and y ordinate can be chosen arbitrarily. Similar remarks hold when the gravity vector is in one of the other coordinate directions. The same holds when using coordinates instead of a grid point.
6. PATM should be equal to the pressure in the air.
7. When coupling surfaces are present then the HYDSTAT ID needs to be referenced by at least one coupling surface. For each coupling surface a different HYDSTAT entry can be defined. Several COUPLE entries can refer to the same HYDSTAT ID. If no HYDSTAT ID is specified on a COUPLE entry then Euler elements associated to this coupling surface will not be initialized with a hydrostatic preset.

**IMPCASE**

Defines a collection of imperfection cases

Defines a collection of imperfection cases by listing of identification numbers of IMPGEOM entries. Used in SOL 400.

Format:

| 1       | 2  | 3       | 4       | 5   | 6       | 7 | 8 | 9 | 10 |
|---------|----|---------|---------|-----|---------|---|---|---|----|
| IMPCASE | ID | IMPFID1 | IMPFID2 | ... | IMPFIDI |   |   |   |    |

Example:

|         |   |   |   |  |  |  |  |  |  |
|---------|---|---|---|--|--|--|--|--|--|
| IMPCASE | 5 | 1 | 2 |  |  |  |  |  |  |
|---------|---|---|---|--|--|--|--|--|--|

**Descriptor**      **Meaning**

ID              Identification number (integer > 0, required)

IMPFIDI      Identification number of imperfection case (integer >0, required)

Remarks:

1. When the ID is referenced by case control IMPERFECT command, the list of IMPGEOM entries are used to apply geometric imperfections.
2. "THRU" can be used to specify IMPFIDI.

**IMPGEOM****Defines Geometric Imperfection**

Defines geometric imperfection by selecting subcases, steps, mode numbers or increments from imperfection input files. Used in SOL 400.

**Format:**

| 1       | 2        | 3     | 4        | 5      | 6      | 7 | 8     | 9 | 10 |
|---------|----------|-------|----------|--------|--------|---|-------|---|----|
| IMPGEOM | IMPFID   |       |          | SETID  | SCALE  |   | UNIT  |   |    |
|         | SUBCASE1 | STEP1 | MODEINC1 | SETID1 | SCALE1 |   | UNIT1 |   |    |
|         | SUBCASE2 | STEP2 | MODEINC2 | SETID2 | SCALE2 |   | UNIT2 |   |    |
|         | etc.     |       |          |        |        |   |       |   |    |

**Example:**

|         |   |   |   |  |       |  |    |  |  |
|---------|---|---|---|--|-------|--|----|--|--|
| IMPGEOM | 1 |   |   |  |       |  |    |  |  |
|         | 1 | 3 | 1 |  | 0.011 |  | 34 |  |  |
|         | 1 | 3 | 2 |  | 0.002 |  | 34 |  |  |

**Descriptor      Meaning**

|                      |   |
|----------------------|---|
| IPMFID               | Identification number of imperfection case (integer >0, required)   |
| SETID                | Default of SETID <sub>i</sub> (integer $\geq 0$ , default=0)  |
| SCALE                | Default of SCALE <sub>i</sub> (real, default=0.0)   |
| UNIT                 | Default of UNIT <sub>i</sub> (integer $\geq 0$ , default=0)   |
| SUBCASE <sub>i</sub> | Identification number of a subcase (integer $\geq 0$ , default=0)   |
| STEP <sub>i</sub>    | Identification number of a step (integer $\geq 0$ , default=0)  |
| MODEINC <sub>i</sub> | Mode number if the selected step by SUBCASE <sub>i</sub> and STEP <sub>i</sub> or subcase by SUBCASE <sub>i</sub> is a result of buckling analysis or modal analysis, increment number if the selected result is from a static analysis. (integer $\geq 0$ , default=0) |
| SETID <sub>i</sub>   | Identification number of a SET1 or SET3 bulk entry (integer $\geq 0$ , default = 0). When it is defined, only the grid points with the id listed in SET1 or SET3 have geometric imperfection effect. If a SET3 is used, it must be GRID type.                           |
| SCALE <sub>i</sub>   | Scale factor (real, default = 0.0)  |
| UNIT <sub>i</sub>    | Imperfection file unit number specified by ASSIGN HDF5IN, INPUTTT2 or IMPFIN FMS command.   |

**Remarks:**

1. All IMPGEOM identification numbers must be unique. An IMPGEOM entry is referenced by IMPERFECT case control directly or by an IMPCASE bulk entry.

2. Geometric imperfection effect is applied to grid points as coordinates variation, the variation is superposed by eigenvectors of multiple modes or displacement of increments defined in multiple continuation lines.
3. An imperfection input file can be an MSC Nastran HDF5 result file, a Nastran op2 file or a text based file called IMPF. To use it, an ASSIGN HDF5IN, INPUTTT2 or IMPFIN FMS command is required.
4. The imperfection input file of HDF5 or op2 can be a result file of SOL 101, 103, 105 or a SOL 400 with ANALYSIS type of BUCK, MODES, STATIC, NLSTATIC or NLTRAN, and must contain eigenvectors or displacement results.
5. IMPF file format is described in remark 11. It can be DISP format or GEOM format. If an IMPF file is used in an IMPGEOM entry, only one continuation line of IMPGEOM is allowed, and SUBCASEi, STEPi and MODEINCi are ignored. If format is GEOM, SCALE and SCALEi are ignored too.
6. SUBCASEi must be specified if the imperfection input file is an op2 or hdf5 file.
7. STEPi is ignored when the imperfection input file is a result of SOL 101, 103 or 105. It must be specified for a SOL 400 result file.
8. MODEINCi is ignored when the file is a result of SOL 101 or a linear static step of SOL 400. It is required for other cases if the imperfection file is an hdf5 or op2 file.
9. UNITi must be specified if UNIT is 0 or not specified.
10. Geometric imperfection is applied to residual structure for superelement models. In the case of part superelement, it is advised to adjust boundary grid point searching tolerance due to geometry changing. This can be done by adding or modifying SEBULK or SECONCT entries.
11. Format of IMPF file

The IMPF file is a csv-like file, delimiters of a line can be spaces, a comma or a tab space. A line starting with a “\$” or “#” is a comment line. If the first line is a comment line and contains string “GEOM” or “GRID”, then it is a GEOM file, i.e., the grid coordinates provided here will replace the grid coordinates in the FE model; otherwise it is a DISP file, i.e., the displacements provided in the file will be superimposed to the coordinates of the corresponding grids.

For each line, the first column is grid point id, followed by x,y and z. For DISP format, x,y and z are coordinate variations from the original coordinates, and coordinate system is MSC Nastran global system, this means that x,y,z are in the coordinate system indicated by CD field of GRID entries in the original input file. For GEOM format, x,y and z are “imperfect” coordinates of the grid points, coordinate system is MSC Nastran input system, this means the values of x,y,z respect CP field of the GRID entries in the original input file.

Below is a DISP format of IMPF file.

disp\_1.impf

---

|      |                      |                       |                     |
|------|----------------------|-----------------------|---------------------|
| 18 , | 0.000000000000E+00 , | 0.000000000000E+00 ,  | 0.000000000000E+00  |
| 19 , | 0.000000000000E+00 , | 0.000000000000E+00 ,  | 0.000000000000E+00  |
| 20 , | 1.882586315422E-04 , | -1.382379038267E-03 , | -5.111620104649E-05 |

```

21 ,   3.080245224512E-04 ,  -1.360710847814E-03 ,  -5.111620104647E-05
22 ,   4.254461607540E-04 ,  -1.328686826189E-03 ,  -5.111620104646E-05
23 ,   5.396298968822E-04 ,  -1.286550695534E-03 ,  -5.111620104647E-05
24 ,   6.497067236577E-04 ,  -1.234623137246E-03 ,  -5.111620104650E-05
25 ,   7.548388899626E-04 ,  -1.173299351397E-03 ,  -5.111620104657E-05
26 ,   8.542262765314E-04 ,  -1.103046049026E-03 ,  -5.111620104668E-05
27 ,   9.471124853400E-04 ,  -1.024397900185E-03 ,  -5.111620104681E-05
28 ,   1.032790596252E-03 ,  -9.379534647748E-04 ,  -5.111620104691E-05
29 ,   1.110608547111E-03 ,  -8.443706371464E-04 ,  -5.111620104698E-05
30 ,   1.179974096322E-03 ,  -7.443616391243E-04 ,  -5.111620104702E-05
31 ,   1.240359330172E-03 ,  -6.386875995705E-04 ,  -5.111620104705E-05

```

Below is a GEOM format of IMPF file.

`geom_example.impf`

---

```

# GEOM
1,   0.000000000000E+00,   0.000000000000E+00,   0.000000000000E+00
2,   1.000000014901E-01,   4.367130542590E-24,   1.464331138939E-08
3,   2.000000029802E-01,   7.552163438029E-24,   3.787209513278E-08
4,   2.999999821186E-01,   8.873575793200E-24,   5.096678857312E-08
5,   3.999999463558E-01,   0.000000000000E+00,   0.000000000000E+00
6,   4.999999701977E-01,   0.000000000000E+00,   0.000000000000E+00
7,   5.999999642372E-01,   0.000000000000E+00,   0.000000000000E+00
8,   6.999999880791E-01,   0.000000000000E+00,   0.000000000000E+00
9,   7.9999998927116E-01,   0.000000000000E+00,   0.000000000000E+00
10,   8.999999165535E-01,   0.000000000000E+00,   0.000000000000E+00
11,   9.999999403954E-01,   0.000000000000E+00,   0.000000000000E+00
12,   1.099999904633E+00,   0.000000000000E+00,   0.000000000000E+00
13,   1.199999928474E+00,   0.000000000000E+00,   0.000000000000E+00
14,   1.299999952316E+00,   0.000000000000E+00,   0.000000000000E+00

```

**INCLUDE****Insert External File**

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

INCLUDE'filename'

**Describer:**

|          |   |
|----------|---|
| filename | Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks. |
|----------|---|

**Example:**

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

/dir123/dir456/dir789/filename.dat

may be included with the following input:

```
INCLUDE '/dir123
          /dir456
          /dir789/filename.dat'
```

3. See the [MSC Nastran 2020 Installation and Operations Guide](#) for more examples.

**INFLCG**

## Airbag Cold Gas Inflator Model

Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

| 1      | 2      | 3       | 4        | 5        | 6       | 7                  | 8     | 9 | 10 |
|--------|--------|---------|----------|----------|---------|--------------------|-------|---|----|
| INFLCG | INFLID | TANKVOL | INITPRES | INITTEMP | INITMAS | $\gamma$<br>GASNAM | $C_v$ | R |    |
|        | $C_p$  |         |          |          |         |                    |       |   |    |

Example:

|        |    |       |        |     |      |     |  |     |  |
|--------|----|-------|--------|-----|------|-----|--|-----|--|
| INFLCG | 11 | 0.875 | 131325 | 293 | 1.37 | 1.4 |  | 286 |  |
|        |    |       |        |     |      |     |  |     |  |

| Descriptor        | Meaning  |
|-------------------|--|
| INFLID            | Unique number of an INFLCG entry. Referenced from ABINFL. (Integer > 0; Required)                                      |
| TANKVOL           | Tank Volume. (Real > 0; Required)  |
| INITPRES          | Initial tank pressure. (Real > 0; Required)  |
| INITTEMP          | Initial tank temperature. (Real > 0; Required)   |
| INITMAS           | Initial gas mass of inflator. (Real > 0; Required)   |
| $\gamma$ , GASNAM | Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or 1) |
| $C_v$             | Specific heat at constant volume. See Remark 6. (Real > 0)   |
| $R$               | Gas constant. See Remark 6. (Real > 0)   |
| $C_p$             | Specific heat at constant pressure. See Remark 6. (Real > 0)   |

Remarks:

1. The INFLCG entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either INITPRESS or INITMASS has to be specified, but not both. The relation between INITMASS and INITPRESS is given by

$$\text{INITPRES} = R \frac{\text{INITMAS}}{\text{TANKVOL}} \text{INITTEMP}$$

4. The cold gas inflator is a reservoir filled with high pressure gas. It is assumed that the volume stays constant at TANKVOL. The mass inside the inflator will steadily decrease due to flow into the Euler domain or to a GBAG. Due to inertia it can happen that the pressure of the inflator becomes less than the outside pressure. In that case some inflow into the inflator occurs. Transport between inflator and the Euler domain or GBAG is based on the constancy of total temperature. This is equivalent to the pressure method.
5. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
6. Specify only two of the four gas constants. They are related as:

$$\nu = \frac{c_p}{c_v} \quad R = c_p - c_v$$

**INFLFRC****Hybrid Inflator Gas Fraction Definition**

Defines the gas fractions as a function of time for hybrid inflators. Used in SOL 700 only.

**Format:**

| 1       | 2      | 3     | 4     | 5     | 6     | 7    | 8 | 9 | 10 |
|---------|--------|-------|-------|-------|-------|------|---|---|----|
| INFLFRC | FRACID | TYPE  |       |       |       |      |   |   |    |
|         | "TIME" | TIME1 | FRAC1 | FRAC2 | FRAC3 | etc. |   |   |    |
|         | "TIME" | TIME2 | FRAC1 | FRAC2 | FRAC3 | etc. |   |   |    |

**Example:**

|         |        |         |        |        |        |        |        |        |  |
|---------|--------|---------|--------|--------|--------|--------|--------|--------|--|
| INFLFRC | 12     |         |        |        |        |        |        |        |  |
|         | TIME   | 14.0E-3 | 11.342 | 13.391 | 9.626  | 57.019 | 7.31   | 1.312  |  |
|         | 0.0    | 0.1     | 0.2    |        |        |        |        |        |  |
|         | TIME   | 14.1E-3 | 43.332 | 6.2817 | 4.5155 | 26.747 | 3.4291 | 0.4898 |  |
|         | 15.077 | 0.0     | 0.4    |        |        |        |        |        |  |

| Descriptor | Meaning  |
|------------|--|
| FRACID     | Unique number of an INFLFRC entry.(Integer > 0, required.)                                   |
| TYPE       | Specifies whether mass fractions or molar fractions will be given (Character, default=MASS.) |
|            | MASS The fractions on INFLFRC are mass fractions.  |
|            | MOLAR The fractions on INFLFRC are molar fractions. See Remark 5.                            |
| TIMEID     | Defines a new line of data (Character, required.)  |
|            | TIME Specifies that data for a new time increment will be given. See Remark 3.               |
| TIMEi      | Time for which the gas fractions are given. (Real $\geq$ 0.0, required.)                     |
| FRACi      | Fraction of gas i at the specified time..See Remark 7. (Real $\geq$ 0.0, required.)          |

**Remarks:**

1. The INFLFRC entry is referenced with FRACID from an AIRBAG entry with the option "INFLATOR" or from INFLHB.
2. Fraction values of the inflowing gas will be linearly interpolated between the specified time increments.
3. Use as many continuation lines as necessary to completely define the gas fractions. The data for a time step are preceded by a TIME keyword. Missing entries will be set to 0.0.

4. The order of the gases for which the fractions are specified is identical to the order in which the gases are specified on the AIRBAG entry with the option “INFLATOR”.
5. At least one line of gas fractions must be given.
6. If molar fractions (TYPE=MOLAR) are to be used, the universal gas constant must be specified through PARAM, UGASC.
7. At least one of the fractions for each time step must have a value greater than 0.0.
8. Fractions for each timestep should add up to 1.0. If this is not the case, they will be scaled so that they do.

**INFLGAS****Inflator Gas Definition**

Defines a thermically ideal gas to be used with a standard or hybrid inflator. Used in SOL 700 only.

**Format:**

| 1       | 2      | 3    | 4     | 5     | 6  | 7  | 8  | 9  | 10 |
|---------|--------|------|-------|-------|----|----|----|----|----|
| INFLGAS | GASNAM | TYPE | VALUE | CPGAS | V1 | V2 | V3 | V4 |    |

**Example:**

|         |     |       |      |          |      |  |  |  |  |
|---------|-----|-------|------|----------|------|--|--|--|--|
| INFLGAS | 101 | RSPEC | 189. | CONSTANT | 846. |  |  |  |  |
|---------|-----|-------|------|----------|------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| GASNAM     | Unique number of an INFLGAS entry. (Integer; Required)  |
| TYPE       | Specific gas constant or molar weight specified. (Character; RSPEC)   |
|            | RSPEC      Specific gas constant  |
|            | MOLWT      Molar weight, see Remark 2.  |
| VALUE      | Value of the variable TYPE. (Real > 0; Required)  |
| CPGAS      | The variation of the specific heat constant at constant pressure. (Character; CONSTANT)   |
|            | CONSTANT      The specific heat is constant and specified in V1.  |
|            | TABLE      The specific heat constant is temperature-dependent. V1 is the number of a TABLED1 entry giving the variation of the specific heat with the temperature. |
| POLY       | The specific heat constant is temperature-dependent. V1 through V4 are the coefficients of a polynomial expression, see Remark 3.                                   |
| V1         | The specific heat constant, the number of a TABLED1 entry or the first polynomial coefficient, depending on the value of CPGAS. (Real or Integer > 0; Required)     |
| V2, V3, V4 | Coefficients of polynomial expression when CPGAS equals POLY. (Real; 0.0)   |

**Remarks:**

1. INFLGAS can be referenced by an INFLTR, INFLHB or INITGAS entry.
2. When the molar weight is given, the universal gas constant  $R_{uni}$  must be specified using PARAM, UGASC, so that:

$$R_{spec} = R_{uni}/MOLWT$$

3. A polynomial expression for  $c_p$  is given by:

$$c_p(t) = V1 + V2 \cdot T + V3 \cdot T^2 + \frac{V4}{T^2}$$

4. The specific heat constant at constant volume  $c_v$  is calculated from the specific heat constant at constant pressure  $c_p$ , the universal gas constant and the molecular weight according to:

$$c_v = c_p(T) - R_{spec}$$

5. The ratio of specific heats is given as:

$$\gamma = c_p / c_v$$

**INFLHB****Hybrid Inflator Model**

Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

**Format:**

| 1      | 2       | 3       | 4       | 5      | 6    | 7 | 8 | 9 | 10 |
|--------|---------|---------|---------|--------|------|---|---|---|----|
| INFLHB | INFLHID | MASFLRT | TBMPT   | TEMPC  | FRAC |   |   |   |    |
|        | GASNAM1 | GASNAM2 | GASNAM3 | -etc.- |      |   |   |   |    |

**Example:**

|        |    |    |   |      |    |  |  |  |  |
|--------|----|----|---|------|----|--|--|--|--|
| INFLHB | 9  | 15 |   | 650. | 12 |  |  |  |  |
|        | 22 | 25 | 3 |      |    |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| INFLID     | Unique number of an INFLHB entry. (Integer > 0; Required)   |
| MASFLRT    | Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)                               |
| TEMPT      | Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0) |
| TEMPC      | Constant value of the temperature of the inflowing gas. See Remark 3. (Real > 0.0)  |
| FRAC       | Number of an INFLFRC entry specifying the fractions of the inflowing gas as a function of time. (Integer > 0; Required)                   |
| GASNAMi    | ID of an INFLGAS entry. See Remark 4. (Integer)   |

**Remarks:**

1. The INFLHB entry can be referenced from a [ABINFL, 1186](#) entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either TEMPC or TEMPT must be specified. The INFLHB entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition as opposed to the dynamic temperature that refers to the temperature of the moving gas.
4. At least one inflator gas must be specified using an INFLGAS entry. There is no limit to the number of inflator gases per INFLHB.

**INFLTNK****Airbag Tanktest Inflator Model**

Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

**Format:**

| 1       | 2        | 3        | 4       | 5       | 6       | 7        | 8        | 9        | 10 |
|---------|----------|----------|---------|---------|---------|----------|----------|----------|----|
| INFLTNK | INFLID   | METHOD   | TPTABLE | TANKVOL | INFLMAS | INITPRES | ENDTPRES | INITTEMP |    |
|         | ENDTEMP  | $\gamma$ | $c_v$   | $R$     | $c_p$   | IPTABLE  | INFLPRES | INFLTEMP |    |
|         | INFLAREA | SFTP     | SFIP    |         |         |          |          |          |    |

**Example:**

|         |     |        |     |      |      |     |  |  |  |
|---------|-----|--------|-----|------|------|-----|--|--|--|
| INFLTNK | 111 | AVTEMP | 10  | 0.12 | 0.01 | 0.0 |  |  |  |
|         |     |        | 1.4 |      | 286. |     |  |  |  |
|         |     |        |     |      |      |     |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| INFLID     | Unique number of an INFLTNK entry. (Integer > 0; Required)  |
| METHOD     | Method of calculating the mass-flowrate: (Character; Required)  |
| AVTEMP     | Average Temperature Method  |
| INFPRES    | Inflator Pressure Method  |
| TPTABLE    | Table number of a TABLED1 entry specifying the tank pressure as a function of time. (Integer > 0; Required)                   |
| TANKVOL    | Tank Volume. (Real > 0; Required)   |
| INFLMAS    | Total gas mass generated by inflator. (Real > 0; Required)  |
| INITPRES   | Initial tank pressure. See Remark 3. (Real > 0; Required)   |
| ENDPRES    | End tank pressure. See Remarks 4. and 5. (Real > 0; Required)   |
| INITTEMP   | Initial tank temperature. See Remark 5. (Real > 0; Required)  |
| ENDTEMP    | End tank temperature. See Remark 5. (Real > 0; Required)  |
| $\gamma$   | Ratio of specific heat constants. See Remark 7. (Real > 0)  |
| $c_v$      | Specific heat at constant volume. See Remark 7. (Real > 0)  |
| $R$        | Gas Constant. See Remark 7. (Real > 0; Required)  |
| $c_p$      | Specific heat at constant pressure. See Remark 7. (Real > 0)  |
| IPTABLE    | Table number of a TABLED1 entry specifying the inflator pressure as a function of time. See Remark 5. (Integer > 0; Required) |
| INFLPRES   | Initial inflator pressure. See Remarks 5. and 6. (Real > 0; Required)   |

| Descriptor | Meaning  |   |
|------------|--|---|
| INFLTEMP   | Temperature of inflowing gas: See Remark 5. (Real > 0 or Character; ATM) |   |
|            | ATM  | Use average temperature of AVTEMP method. |
|            | Real value   | User specified temperature.               |
| INFLAREA   | Total area of inflator holes. See Remark 5. (Real > 0; Required)         |   |
| SFTP       | Scale factor for tank pressure. See Remark 5. (Real > 0; 1.0)            |   |
| SFIP       | Scale factor for inflator pressure. See Remark 5. (Real > 0, 1.)         |   |

**Remarks:**

1. The INFLTNK entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. The initial tank pressure entry (INITPRES) is interpreted as an absolute pressure and used to define reference pressure at t=0 in the tank. The difference between INITPRES and the pressure value at t=0 from the table will be added to the entire pressure curve of TPTABLE.
4. The end tank pressure entry (ENDPRES) is interpreted as an absolute pressure at t=tend of tank pressure table (TPTABLE). This value is used for calculation of total generated mass in the tank.
5. This field must be specified only when Inflator Pressure Method (INFPRES) is defined in the METHOD field.
6. The initial inflator pressure entry (INFLPRES) is interpreted as an absolute pressure and used to define reference pressure at t=0 in the inflator. The difference between INFLPRES and the pressure value at t=0 from the table will be added to the entire pressure curve of IPTABLE.
7. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

**INFLTR****Airbag Inflator Model**

Defines the inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

**Format:**

| 1      | 2      | 3       | 4     | 5     | 6                   | 7     | 8   | 9     | 10 |
|--------|--------|---------|-------|-------|---------------------|-------|-----|-------|----|
| INFLTR | INFLID | MASFLRT | TEMPT | TEMPC | $\gamma$<br>GASNAME | $c_v$ | $R$ | $c_p$ |    |
|        | MID    |         |       |       |                     |       |     |       |    |

**Example:**

|        |   |     |   |       |     |       |  |  |  |
|--------|---|-----|---|-------|-----|-------|--|--|--|
| INFLTR | 5 | 100 |   | 907.0 | 1.5 | 283.0 |  |  |  |
|        | 0 | 0   | 0 |       |     |       |  |  |  |

| Descriptor         | Meaning   |
|--------------------|---|
| INFLID             | Unique number of an INFLTR entry. (Integer > 0; Required)   |
| MASFLRT            | Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)                               |
| TEMPT              | Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0) |
| TEMPC              | Constant value of the static temperature of the inflowing gas. See Remark 3. (Real > 0)   |
| $\gamma$ , GASNAME | Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or Character)            |
| $c_v$              | Specific heat at constant volume. See Remark 5. (Real > 0)  |
| $R$                | Gas constant. See Remark 5. (Real > 0)  |
|                    | Specific heat at constant pressure. See Remark 5. (Real > 0)  |
| MID                | Material MATDEUL ID of the Inflator material. See Remark 2. Only used for MMHYDRO solver. (Integer $\geq 0$ )                             |

### Remarks:

1. The INFLTR entry can be referenced from an ABINFL entry.
2. When used in combination with the single material hydrodynamic Euler solver an EOSGAM (ideal gas) equation of state is required. In that case the material number MID can be left blank. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM. The Multi-material solver does not allow the use of gas fractions.

3. Either TEMP-C or TEMP-T must be specified. The INFLTR entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition, as opposed to the dynamic temperature which refers to the temperature of the moving gas.
4. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself, see Remark 5. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
5. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

**INITGAS****Gasbag or Coupling Surface Initial Gas Fraction Definition**

Specifies the initial gas composition inside a gasbag or Euler coupling surface. Used in SOL 700 only.

Format:

| 1       | 2     | 3 | 4       | 5     | 6       | 7     | 8     | 9 | 10 |
|---------|-------|---|---------|-------|---------|-------|-------|---|----|
| INITGAS | INTID |   | GASNAM1 | FRAC1 | GASNAM2 | FRAC2 | -etc. |   |    |

Example:

|         |   |  |    |     |    |      |  |  |  |
|---------|---|--|----|-----|----|------|--|--|--|
| INITGAS | 4 |  | 14 | 0.4 | 32 | 0.11 |  |  |  |
|---------|---|--|----|-----|----|------|--|--|--|

**Descriptor      Meaning**

INTID      Unique number of an INITGAS entry. (Integer > 0; Required)

GASNAMi      ID of an INFLGAS entry. See Remark 3. (Integer > 0)

FRACi      Mass fraction of gas i. See Remark 4. (Real  $\geq 0.0$ )

Remarks:

1. The INITGAS entry can be used to specify the initial gas composition for a gasbag or for an Eulerian coupling surface. The INTID must be referenced either from a GBAG cad or a COUPLE entry.
2. Use as many continuation lines as necessary to completely define the gas fractions.
3. At least one INFLGAS reference must be given.
4. Fractions should add up to 1.0. If this is not the case, they will be scaled so that they do.

**IPSTRAIN**

## Initial Equivalent Plastic Strain Values in SOL 400

Defines initial equivalent plastic strain values. This is the initial plastic strain option used in SOL 400 only.  
 (NOTE: This entry is SINGLE FIELD ONLY! Double field will fatal)

Format:

| 1        | 2    | 3    | 4    | 5    | 6    | 7    | 8      | 9 | 10 |
|----------|------|------|------|------|------|------|--------|---|----|
| IPSTRAIN | EID1 | EID2 | INT1 | INTN | LAY1 | LAYN | STRAIN |   |    |

Example:

|          |      |      |   |   |   |   |       |  |  |
|----------|------|------|---|---|---|---|-------|--|--|
| IPSTRAIN | 2001 | 2020 | 1 | 4 | 1 | 5 | 0.025 |  |  |
|----------|------|------|---|---|---|---|-------|--|--|

**Descriptor**      **Meaning**

|        |   |
|--------|---|
| EID1   | First Element ID to which these strains apply. (Integer > 0)  |
| EID2   | Last Element ID to which these strains apply. (Integer; Default = EID1)   |
| INT1   | First Integration point for which the strain applies. (Integer > 0; Default = 1)  |
| INTN   | Last Integration point for which the strain applies. (Integer > 0; Default = 4)   |
| LAY1   | First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.) |
| LAYN   | Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)  |
| STRAIN | Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)  |

Remarks:

1. This entry only applies to SOL 400 advanced nonlinear elements (selected with PSHLN1, PSLDN1, etc.) and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.

**IPSTRN****Initial Equivalent Plastic Strain Values in SOL 600**

Defines initial equivalent plastic strain values. This is the MSC Marc's initial plastic strain option used in SOL 600 only.

**Format:**

| 1      | 2    | 3    | 4    | 5    | 6    | 7    | 8      | 9 | 10 |
|--------|------|------|------|------|------|------|--------|---|----|
| IPSTRN | EID1 | EID2 | INT1 | INTN | LAY1 | LAYN | STRAIN |   |    |

**Example:**

|        |      |      |   |   |   |   |       |  |  |
|--------|------|------|---|---|---|---|-------|--|--|
| IPSTRN | 2001 | 2020 | 1 | 4 | 1 | 5 | 0.025 |  |  |
|--------|------|------|---|---|---|---|-------|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID1       | First Element ID to which these strains apply. (Integer > 0)  |
| EID2       | Last Element ID to which these strains apply. (Integer; Default = EID1)   |
| INT1       | First Integration point for which the strain applies. (Integer > 0; Default = 1)  |
| INTN       | Last Integration point for which the strain applies. (Integer > 0; Default = 4)   |
| LAY1       | First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.) |
| LAYN       | Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)  |
| STRAIN     | Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)  |

**Remarks:**

1. This entry only applies to SOL 600 advanced nonlinear elements and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.
3. For SOL 600 only, if the extra precision available with large field formats is not required and small field precision is adequate, the header IPSTRAIN may be used instead of IPSTRN.

**ISTRESS****Initial Stress Values in SOL 400 and SOL 600**

Defines initial stress values. This is the MSC Marc's initial stress option used in SOL 400 and SOL 600 only.

**Format:**

| 1       | 2       | 3       | 4       | 5       | 6       | 7    | 8       | 9       | 10 |
|---------|---------|---------|---------|---------|---------|------|---------|---------|----|
| ISTRESS | EID1    | EID2    | INT1    | INTN    | LAY1    | LAYN | STRESS1 | STRESS2 |    |
|         | STRESS3 | STRESS4 | STRESS5 | STRESS6 | STRESS7 |      |         |         |    |

**Example:**

|         |      |       |     |       |   |   |        |        |  |
|---------|------|-------|-----|-------|---|---|--------|--------|--|
| ISTRESS | 2001 | 2020  | 1   | 4     | 1 | 5 | 45000. | -2000. |  |
|         | 0.0  | 4500. | 0.0 | 2350. |   |   |        |        |  |

| Descriptor | Meaning   |
|------------|---|
| EID1       | First Element ID to which these stresses apply. (Integer > 0)   |
| EID2       | Last Element ID to which these stresses apply. (Integer; Default = EID1)  |
| INT1       | First Integration point for which the stress applies. (Integer > 0; Default = 1)  |
| INTN       | Last Integration point for which the stress applies. (Integer > 0; Default = 4)   |
| LAY1       | First element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.) |
| LAYN       | Last element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)  |
| STRESS(i)  | Up to 7 stress components may be entered. (Real; Default = 0.0)   |

**Remarks:**

1. This entry only applies when MSC Marc is executed from Nastran using SOLs 400 and 600 and is ignored for other solutions.
2. Initial stresses must be self-equilibrating and may not produce material nonlinearity.
3. Stress components are as follows:

Definitions:

s - normal type stress

t - shear type stress

x,y,z in global system

1,2,3 in element local system

3D solid elements (for example type 7)

1 - sxx

2 - syy

3 - szz

4 - txy

5 - tyz

6 - tzx

7 - hydrostatic pressure (Herrmann elements only, otherwise 7 should be blank)

Thick shells (for example type 75)

1 - s11

2 - s22

3 - t12

4 - t23

5 - t31

Thin shells (for example type 72)

1 - s11

2 - s22

3 - t12

Beams (for example type 14 or 98)

1 - s - axial

2 - t - twist

**ITER****Iterative Solver Options**

Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153 and 400.

**Format:**

| 1    | 2   | 3             | 4             | 5      | 6 | 7 | 8 | 9 | 10 |
|------|-----|---------------|---------------|--------|---|---|---|---|----|
| ITER | SID |               |               |        |   |   |   |   |    |
|      |     | OPTION1VALUE1 | OPTION2VALUE2 | -etc.- |   |   |   |   |    |

**Example:**

|      |     |   |  |
|------|-----|---|--|
| ITER | 100 |   |  |
|      |     | ITSEPS=1.0E0-7, MSGFLG=YES, PRECOND=BICWELL, IPAD=3 |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0).   |
| PRECOND    | Preconditioner option. (Character; Default = "BIC" for real analysis, "BICCMPLX" for complex analysis and "PBDJ" for p-version analysis.) See Remarks 3. and 4.   |
| J          | Jacobi  |
| JS         | Jacobi with diagonal scaling.   |
| C          | Incomplete Cholesky.  |
| CS         | Incomplete Cholesky with diagonal scaling.  |
| RIC        | Reduced incomplete Cholesky.  |
| RICS       | Reduced incomplete Cholesky with diagonal scaling.  |
| PBCJ       | p-version block Cholesky/Jacobi.  |
| PBDJ       | p-version block Direct/Jacobi.  |
| PBDC       | p-version block Direct/Cholesky.  |
| BIC        | Block incomplete Cholesky for real problems.  |
| BICCMPLX   | Block incomplete Cholesky for complex problems.   |
| CASI       | Element-based third party iterative solver.   |
| USER       | User given preconditioning. For direct frequency response (SOL 108): a decomposition will be done for 1st frequency and the factor will be used for all subsequent frequencies as a preconditioner with the iterative solver. Other solutions require a DMAP alter. Please refer to the <i>MSC Nastran Numerical Methods User's Guide</i> description of the SOLVIT module. |
| CONV       | Convergence criterion. (Character; Default = "AREX")  |

| Descriptor | Meaning   |  |
|------------|---|--|
| AR         | $\ r\ /\ b\ $ where r is the residual vector of current iteration and b is the initial load vector; internal criterion.   |  |
| GE         | Alternative convergence criterion using geometric progression and the differences between two consecutive solution updates; internal criterion.   |  |
| AREX       | Same criterion as AR but with the additional consideration of the external convergence criterion. See Remark 2. (Default).  |  |
| GEXX       | Same criterion as GE but with the additional consideration of the external convergence criterion. See Remark 2.   |  |
| MSGFLG     | Message flag. (Character; Default = "NO")   |  |
| YES        | Messages will be printed for each iteration.  |  |
| NO         | Only minimal messages will be printed from the iterative solver (Default).  |  |
| ITSEPS     | User-given convergence parameter epsilon. (Real > 0.0)<br>Default= 1.E-4 for PRECOND = "CASI"<br>Default= 1.E-8 for PRECOND = "CASI" with contact in the model<br>Default= 1.E-6 for all other PRECOND options  |  |
| ITSMAX     | Maximum number of iterations. (Integer > 0; Default = N/4 where N is the number of rows in the matrix)  |  |
| IPAD       | Padding value for RIC, RICS, BIC, and BICCMPLX preconditioning. (Integer > 0)<br>Default = 0 for PRECOND = "RIC" or "RICS"<br>Default = 2 for PRECOND = "BIC" for purely three-dimensional models and three for two-dimensional and mixed element models. IPAD may be reset automatically by the program to the best value.<br>Default = 5 for PRECOND = "BICCMPLX".  |  |
| IEXT       | Extraction level in reduced incomplete Cholesky preconditioning. Block structuring method in block incomplete Cholesky preconditioning. (Integer = 0 thought 7; Default = 0)<br>0 Uses USET/SIL tables (Default).<br>1 - 7 The default value of 0 is recommended for all problems. The values 1 - 7 use a heuristic algorithm with a maximum block size equal to IEXT. Although setting IEXT to a value other than 0 could lead to slightly improved performance or reduced disk space use, it should be considered exploratory without the expectation of a benefit. |  |
| PREFONLY   | Specifies early termination of the iterative solver. (Integer = 0 or -1; Default = 0)<br>0 Runs to completion (Default).<br>-1 Terminates after preface giving resource estimates.  |  |

**Remarks:**

1. The optional ITER Bulk Data entry is selected by the SMETHOD Case Control command and is only required to override the defaults specified above.
2. The external epsilon is computed as follows:

$$\varepsilon = \frac{(r, x)}{(b, x)}$$

where r is the final residual vector, x is the final solution vector and b is the initial load vector ( $r, x$ ) indicates the inner product of  $r$  and  $x$  and  $(b, x)$  indicates the inner product of  $b$  and  $x$ .

3. See the *MSC Nastran Numerical Methods User's Guide* for more information on these options.
4. The element-based iterative solver is primarily intended for the solution of very large solid element structural analysis problems. The following restrictions apply:
  - SOLs 101, 200 and 400 only. SOL 200 availability is limited to topology optimization and thus cannot be used for COMPLIANCE, FRMASS, WEIGHT, and VOLUME in DRESP1.
  - SOL 101 convergence parameter epsilon of 1.E-4 may be too large for some models and a decrease to 1.E-8 may be necessary.
  - Solver selection criteria and parameters cannot vary across subcases.
  - No GENEL elements allowed
  - x2GG/x2PP direct input matrix selection is allowed; however, the matrix size is limited to 100 grid points and must be symmetric.
  - No ASET/OMIT reduction allowed
  - Inertia relief is supported in SOL 101 for PARAM,INREL,-1. A SUPORT entry is required. PARAM,INREL,-2 is not supported.
  - No transfer functions allowed
  - No RFORCE or PLOADX follower forces allowed
  - Follower force stiffness must be symmetricized
  - No heat transfer allowed
  - No p-elements allowed
  - DMP is not supported with the CASI solver.

Only BAR, BEAM, BUSH, ROD, CONMi, CONROD, DAMPi, ELASi, HEXA, MASSi, PENTA, QUAD4, QUAD8, QUADR, SEAM, SHEAR, TRIA3, TRIA6, TRIAR, TETRA, VISC and WELD elements are allowed.

5. GPGPU devices are not supported for iterative methods.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## LBSH3DG      Tables used by an NLBSH3D to find force as function of displacement, velocity and rotor speed

Defines the force tables of NLBSH3D nonlinear loads as function of relative displacement or velocity and current rotor speed.

Format:

FORM = ASYM (Asymmetric matrix representation)

| 1       | 2   | 3    | 4        | 5        | 6   | 7   | 8 | 9 | 10 |
|---------|-----|------|----------|----------|-----|-----|---|---|----|
| LBSH3DG | LID | FORM | MIN_FREQ | MAX_FREQ |     |     |   |   |    |
|         | P11 | P12  | P13      | P14      | P15 | P16 |   |   |    |
|         | P21 | P22  | P23      | P24      | P25 | P26 |   |   |    |
|         | P31 | P32  | P33      | P34      | P35 | P36 |   |   |    |
|         | P41 | P42  | P43      | P44      | P45 | P46 |   |   |    |
|         | P51 | P52  | P53      | P54      | P55 | P56 |   |   |    |
|         | P61 | P62  | P63      | P64      | P65 | P66 |   |   |    |

FORM = DIAG (Only Diagonal terms representation)

| 1       | 2   | 3    | 4        | 5        | 6   | 7   | 8 | 9 | 10 |
|---------|-----|------|----------|----------|-----|-----|---|---|----|
| LBSH3DG | LID | FORM | MIN_FREQ | MAX_FREQ |     |     |   |   |    |
|         | P11 | P22  | P33      | P44      | P55 | P66 |   |   |    |

FORM = SYM (Symmetric matrix representation)

| 1       | 2   | 3    | 4        | 5        | 6   | 7   | 8 | 9 | 10 |
|---------|-----|------|----------|----------|-----|-----|---|---|----|
| LBSH3DG | LID | FORM | MIN_FREQ | MAX_FREQ |     |     |   |   |    |
|         | P11 |      |          |          |     |     |   |   |    |
|         | P21 | P22  |          |          |     |     |   |   |    |
|         | P31 | P32  | P33      |          |     |     |   |   |    |
|         | P41 | P42  | P43      | P44      |     |     |   |   |    |
|         | P51 | P52  | P53      | P54      | P55 |     |   |   |    |
|         | P61 | P62  | P63      | P64      | P65 | P66 |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| LID        | Load identification group number for a NLBSH3D. (Integer > 0; Required) |
| FORM       | Type of Input Curves. (Character; Default SYM)                          |
| SYM        | Symmetric, only lower triangular terms needed                           |

| Descriptor | Meaning  |
|------------|--|
| DIAG       | Only diagonal terms are needed   |
| ASYM       | Asymmetric, all terms are needed   |
| MIN_FREQ   | Minimum valid rotor speed for all the table lookups provided. (Real). See remark <a href="#">2</a> .   |
| MAX_FREQ   | Max valid rotor speed for all the table lookups provided. (Real > 0.0 Required). Refer remark <a href="#">2</a> .                                      |
| Pij        | TABLED5 Id that provides TABLED1 ids as a function of rotor speed. The TABLED1 will have force as a function of relative displacement and/or velocity. |

#### Remarks:

1. Values for the nonlinear forces will be determined using interpolation based on rotor speed and relative deflection/velocity of grid pair. If the rotor speed is beyond the range specified by MIN\_FREQ and MAX\_FREQ, a FATAL message will be produced. (Extrapolation will NOT be used beyond this range).
2. For SYNC analysis, rotor speed is obtained from NLFREQ / NLFREQ1 bulk data card. For ASYNC analysis, rotor speed is obtained from RGYRO bulk data card.
3. All the tables (TABLED5 Pij) should provide lookup tables (TABLED1) for the range of rotor speeds between Min\_Freq and Max\_Freq.

**LEAKAGE**

## Mass Loss Through Holes or Permeability of the GBAG or COUPLE Surface

Defines the porosity model to be used with GBAG or COUPLE. Used in SOL 700 only.

Format:

| 1       | 2   | 3     | 4     | 5       | 6        | 7     | 8      | 9 | 10 |
|---------|-----|-------|-------|---------|----------|-------|--------|---|----|
| LEAKAGE | LID | PORID | SUBID | PORTYPE | PORTYPID | COEFF | COEFFV |   |    |

Example:

|         |   |     |     |         |    |  |      |  |  |
|---------|---|-----|-----|---------|----|--|------|--|--|
| LEAKAGE | 7 | 100 | 365 | PERMEAB | 63 |  | 0.99 |  |  |
|---------|---|-----|-----|---------|----|--|------|--|--|

| Descriptor | Meaning  |  |
|------------|--|--|
| LID        | Unique number of a LEAKAGE entry. (Integer > 0; Required)  |  |
| PORID      | Number of a set of LEAKAGE entries PORID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)   |  |
| SUBID      | > 0<br>Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface, as defined on the GBAG or COUPLE entry.<br><br>= 0<br>LEAKAGE definitions are used for the entire surface as defined on the GBAG or COUPLE entry.   |  |
| PORTYPE    | Defines the type of porosity. (Character; Required)  |  |
| PORFLOW    | The PORFLOW logic is used to model a constant flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLOW entry, with ID as defined on the PORTYPID.       |  |
| PORFLWT    | The PORFLWT logic is used to model a time dependent flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLWT entry, with ID as defined on the PORTYPID. |  |

| Descriptor | Meaning   |
|------------|---|
| PORHOLE    | The PORHOLE logic can be used to model small holes in an air bag. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORHOLE entry, with ID as defined on the PORTYPID.  |
| PERMEAB    | The PERMEAB logic is used to model permeable air-bag material. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference between the gas inside the air bag and the environmental pressure. The function is specified on a PERMEAB entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV |
| PORFGBG    | The PORFGBG logic can be used to model gas flow through a hole in the coupling surface connected to a GBAG. A BSURF defines the hole. The open area of the hole is equal to the area of the surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFGBG entry, with ID as defined on the PORTYPID.  |
| PERMGBG    | The PERMGBG logic is used to model gas flow through a permeable area in the coupling surface connected to a GBAG. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference. This function is specified on a PERMGBG entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV.              |
| PORFCPL    | The PORFCPL logic can be used to model gas flow through a hole in the coupling surface connected to another coupling surface. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFCPL entry, with ID as defined on the PORTYPID.  |
| PORHYDS    | Prescribes a hydrostatic pressure profile.  |
| PORTYPID   | Porosity ID. References existing PORTYPE entry. (Integer > 0; Required)   |
| COEFF      | Method of defining the porosity coefficient. (Character; CONSTANT)  |
| CONSTANT   | The porosity coefficient is constant and specified on COEFFV.   |

| Descriptor | Meaning  |
|------------|--|
| TABLE      | The porosity coefficient varies with time. COEFV is the number of a TABLED1 entry giving the variation with time.              |
| COEFFV     | The porosity coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or Integer < 0, 1.0) |

#### Remarks:

1. The combination of multiple LEAKAGEs with different PORTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to LEAKAGE) or an Eulerian model (COUPLE to LEAKAGE). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

The options PORFGBG and PERMGBG can be used to model air bags with multiple compartments.

**LOAD****Static Load Combination (Superposition)**

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, DAREA (if these entries have been converted), PLOAD, PLOAD1, PLOAD2, PLOADB3, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries (as well as SPCD and SPCR for SOL 600 only). In addition, all the thermal loads are also supported in the NASTRAN thermal analysis (SOL 153 and SOL 400 with analysis=hstat) such as QBDY1, QBDY2, QBDY3, QVECT, QVOL, SLOAD, QHBDY Bulk Data entries.

**Format:**

| 1    | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|------|-----|----|--------|----|----|----|----|----|----|
| LOAD | SID | S  | S1     | L1 | S2 | L2 | S3 | L3 |    |
|      | S4  | L4 | -etc.- |    |    |    |    |    |    |

**Example:**

|      |     |      |     |   |     |   |  |  |  |
|------|-----|------|-----|---|-----|---|--|--|--|
| LOAD | 101 | -0.5 | 1.0 | 3 | 6.2 | 4 |  |  |  |
|------|-----|------|-----|---|-----|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)                                      |
| S          | Overall scale factor. (Real)   |
| Si         | Scale factor on Li. (Real)   |
| Li         | Load set identification numbers defined on entry types listed above. (Integer > 0) |

**Remarks:**

1. The load vector  $\{P\}$  is defined by

$$\{P\} = S \sum_i Si \{P_{Li}\}$$

2. Load set IDs (Li) must be unique.
3. This entry must be used if acceleration loads (GRAV entry) are to be used with any of the other types.
4. In the static solution sequences, SID must be selected by the LOAD Case Control command. In dynamic solution sequences the EXCITEID of a RLOADi or TLOADi may refer to a LOAD entry.
5. A LOAD may be referenced by another LOAD entry. Nested LOAD references are permitted. A LOAD entry may not reference itself.
6. Note that the LOAD entry is scaled differently when using the RFORCE with rotor dynamics. See Remark 21. of the RFORCE entry.
7. LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. In dynamic analysis refer to Remark 2. of the SPCD, 3127 entry.

8. If Modules are present then this entry may only be specified in the main Bulk Data section.

**LOADCLID****Define Solution Load Combinations Using Part Load ID**

Define a loading combination in PAA using the Load ID from SUBCASEs of the Parts.

**Format:**

| 1         | 2    | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-----------|------|------|---|---|---|---|---|---|----|
| LOADCLID  | LID  | S    |   |   |   |   |   |   |    |
|           | S1   | LID1 |   |   |   |   |   |   |    |
| PARTNAME1 |      |      |   |   |   |   |   |   |    |
|           | S2   | LID2 |   |   |   |   |   |   |    |
| PARTNAME2 |      |      |   |   |   |   |   |   |    |
|           | Etc. |      |   |   |   |   |   |   |    |

**Example:**

|                  |      |      |  |  |  |  |  |  |  |
|------------------|------|------|--|--|--|--|--|--|--|
| LOADCLID         | 202  | 1.00 |  |  |  |  |  |  |  |
|                  | 1.00 | 101  |  |  |  |  |  |  |  |
| first_part_name  |      |      |  |  |  |  |  |  |  |
|                  | 1.0  | 301  |  |  |  |  |  |  |  |
| second_part_name |      |      |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| LID        | Load set ID. Selected by the LOAD Case Control command. (Integer > 0)                    |
| S          | Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)                     |
| Si         | Scale factor applied to Load ID loads. (Real < > 0)                                      |
| LIDi       | Load ID specified during GENERATE by LOAD or CLOAD Case Control directive. (Integer > 0) |
| PARTNAMEi  | Name of Part. (64 -Characters maximum)   |

**Remarks:**

1. LOADCLID entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCLID allows a loading combination to be created using the Load id (LIDi) from case control. In each PAA run, each SUBCASE may have a LIDi provided. These are used by the LOADCLID to define a loading combination.
3. As with other loading entries, all loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

**LOADCNAM****Define Solution Load Combination by Load Name**

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

**Format:**

| 1         | 2    | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-----------|------|---|---|---|---|---|---|---|----|
| LOADCNAM  | LID  | S |   |   |   |   |   |   |    |
|           | S1   |   |   |   |   |   |   |   |    |
| LOADNAME1 |      |   |   |   |   |   |   |   |    |
| PARTNAME1 |      |   |   |   |   |   |   |   |    |
|           | S2   |   |   |   |   |   |   |   |    |
| LOADNAME2 |      |   |   |   |   |   |   |   |    |
| PARTNAME2 |      |   |   |   |   |   |   |   |    |
|           | Etc. |   |   |   |   |   |   |   |    |

**Example:**

|                                    |     |      |  |  |  |  |  |  |  |
|------------------------------------|-----|------|--|--|--|--|--|--|--|
| LOADCNAM                           | 101 | 1.00 |  |  |  |  |  |  |  |
|                                    | 2.5 |      |  |  |  |  |  |  |  |
| gravity_x                          |     |      |  |  |  |  |  |  |  |
| outboard_right_side_engine_nacelle |     |      |  |  |  |  |  |  |  |
|                                    | 1.1 |      |  |  |  |  |  |  |  |
| gravity_x_on_another_part          |     |      |  |  |  |  |  |  |  |
| second_part_name                   |     |      |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| LID        | Load set ID. Selected by the LOAD Case Control command. (Integer > 0) |
| S          | Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < 0)    |
| Si         | Scale factor applied to LOADNAME loads. (Real < 0)                    |
| LOADNAMEi  | Name of Load for that Part. (64 -Characters maximum)                  |
| PARTNAMEi  | Name of Part. (64 -Characters maximum)                                |

**Remarks:**

1. LOADCNAM entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCNAM allows a loading combination to be created using the LOADNAME from case control. In each PAA run, each SUBCASE may have a LOADNAME provided. These LOADNAMEs are used by the LOADCNAM to define a loading combination.
3. All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

**LOADCSUB****Define Solution Load Combinations Using Subcase Number**

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

**Format:**

| 1         | 2    | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-----------|------|------|---|---|---|---|---|---|----|
| LOADCSUB  | LID  | S    |   |   |   |   |   |   |    |
|           | S1   | SUB1 |   |   |   |   |   |   |    |
| PARTNAME1 |      |      |   |   |   |   |   |   |    |
|           | S2   | SUB2 |   |   |   |   |   |   |    |
| PARTNAME2 |      |      |   |   |   |   |   |   |    |
|           | Etc. |      |   |   |   |   |   |   |    |

**Example:**

|                                  |      |     |  |  |  |  |  |  |  |
|----------------------------------|------|-----|--|--|--|--|--|--|--|
| LOADCSUB                         | 1001 | 1.0 |  |  |  |  |  |  |  |
|                                  | 1.0  | 1   |  |  |  |  |  |  |  |
| part_we_are_using_subcase_1_from |      |     |  |  |  |  |  |  |  |
|                                  | 1.0  | 3   |  |  |  |  |  |  |  |
| part_we_are_using_subcase_3_from |      |     |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| LID        | Load set ID. Selected by the LOAD Case Control command. (Integer > 0)  |
| S          | Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)   |
| Si         | Scale factor applied to Load ID loads. (Real < > 0)                    |
| SUBi       | Subcase Number specified during GENERATE or COMBINE run. (Integer > 0) |
| PARTNAMEi  | Name of Part. (Character, C64)   |

**Remarks:**

1. LOADCSUB entries are used only in the COMBINE and SOLVE steps of PAA processing.
2. The LOADCSUB allows a loading combination to be created using the SUBCASE ids from case control. In each PAA run, each SUBCASE has an id. These ids are used by the LOADCSUB to define a loading combination.
3. All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

**LOADCYH****Harmonic Load Input for Cyclic Symmetry**

Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.

**Format:**

| 1       | 2   | 3 | 4   | 5     | 6  | 7  | 8  | 9  | 10 |
|---------|-----|---|-----|-------|----|----|----|----|----|
| LOADCYH | SID | S | HID | HTYPE | S1 | L1 | S2 | L2 |    |

**Example:**

|         |    |     |   |   |     |    |  |  |  |
|---------|----|-----|---|---|-----|----|--|--|--|
| LOADCYH | 10 | 1.0 | 7 | C | 0.5 | 15 |  |  |  |
|---------|----|-----|---|---|-----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)                                      |
| S          | Scale Factor. (Real)   |
| HID        | Harmonic. See Remark 6. (Integer or blank)   |
| HTYPE      | Harmonic type. (Character: "C", "S", "CSTAR" "SSTAR", "GRAV", "RFORCE", or blank). |
| Si         | Scale factor on Li. (Real)   |
| Li         | Load set identification number. See Remark 10. (Integer > 0)                       |

**Remarks:**

1. The LOADCYH entry is selected with the Case Control command LOAD = SID.
2. If HTYPE is blank, the load will be applied to all applicable types in the problem.
3. If HTYPE is "GRAV" or "RFORCE", GRAV or RFORCE entry loading will be used. Harmonic loads for appropriate available harmonics will be generated automatically in these cases.
4. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
5. If L1 refers to a set ID defined by an SPCD entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX, and SLOAD or converted DAREA entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
6. If HTYPE is "GRAV" or "RFORCE", the entry in HID will be ignored and therefore may be blank. S2 and L2 must be blank for this case.
7. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYN entries.
8. If HTYPE = "C", "S", "CSTAR", or "SSTAR", the load on component (HTYPE) of harmonic (HID) is  $L = S(S1 \cdot L1 + S2 \cdot L2)$ .

9. S must be nonzero. In addition, either S1 or S2 must be nonzero.
10. L1 and L2 may reference any of the static or dynamic loading entries including GRAV and RFORCE.

**LOADCYN****Physical Load Input for Cyclic Symmetry**

Defines a physical static or dynamic load for use in cyclic symmetry analysis.

**Format:**

| 1       | 2   | 3 | 4     | 5      | 6  | 7  | 8  | 9  | 10 |
|---------|-----|---|-------|--------|----|----|----|----|----|
| LOADCYN | SID | S | SEGID | SEGTYP | S1 | L1 | S2 | L2 |    |

**Example:**

|         |    |     |   |   |     |    |  |  |  |
|---------|----|-----|---|---|-----|----|--|--|--|
| LOADCYN | 10 | 1.0 | 1 | R | 0.5 | 17 |  |  |  |
|---------|----|-----|---|---|-----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)    |
| S          | Scale Factor. (Real)                             |
| SEGID      | Segment identification number. (Integer)         |
| SEGTYP     | Segment type. (Character: "R", "L", or blank)    |
| Si         | Scale Factors. (Real)                            |
| Li         | Load set ID numbers. See Remark 8. (Integer > 0) |

**Remarks:**

1. The LOADCYN entry is selected by the LOAD Case Control command.
2. If SEGTYP is blank, both R and L segments will be used in DIH-type symmetry.
3. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
4. If L1 refers to a set ID defined by SPCD loading entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
5. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYH entries.
6. The load on the segment (or half-segment) is  $L = S(S1 \cdot L1 + S2 \cdot L2)$ .
7. S must be nonzero. In addition, either S1 or S2 must be nonzero.
8. L1 and L2 may reference any of the static or dynamic loading entries except GRAV and RFORCE.
9. For cyclic buckling loads may only be applied to the first segment and only zero harmonic loads may be applied so the LOADCYH entry should be used.

**LOADCYT**

## Table Load Input for Cyclic Symmetry

Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.

**Format:**

| 1       | 2   | 3        | 4        | 5       | 6        | 7        | 8       | 9 | 10 |
|---------|-----|----------|----------|---------|----------|----------|---------|---|----|
| LOADCYT | SID | TABLEID1 | LOADSET1 | METHOD1 | TABLEID2 | LOADSET2 | METHOD2 |   |    |

**Example:**

|         |    |    |    |  |    |    |   |  |  |
|---------|----|----|----|--|----|----|---|--|--|
| LOADCYT | 10 | 19 | 27 |  | 21 | 26 | 1 |  |  |
|---------|----|----|----|--|----|----|---|--|--|

| Descriptor           | Meaning  |
|----------------------|--|
| SID                  | Load set identification number. (Integer > 0)  |
| TABLEID <sub>i</sub> | Table ID for table load input for load set L <sub>i</sub> . See Remark 3. (Integer > 0)                                      |
| LOADSET <sub>i</sub> | Load set L <sub>i</sub> . (Integer > 0)  |
| METHOD <sub>i</sub>  | Method of interpolation. See Remark 5. (Integer: 0 or 1; Default = 0)  |
| 0                    | Interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. (Default) |
| 1                    | Interpolate the magnitude of the load at corresponding grid points in all segments.  |

**Remarks:**

1. The LOADCYT entry is selected by the LOAD Case Control command.
2. The load set ID given in fields 4 or 7 of this entry may refer to FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, SPCD, TEMP, or TEMPP1 Bulk Data entries.
3. Either TABLED1 or TABLED2 type tabular data of azimuth angle (X<sub>i</sub>) versus scale factors (Y<sub>i</sub>) may be used. The azimuth angle values must be in degrees.
4. The scale factors given in the tables referenced by TABLED<sub>i</sub> entries will be applied only to the magnitudes of the loads defined by LOADSET IDs given in fields 4 or 7.
5. For grid point loading entries, (like FORCE, MOMENT, SPCD, and TEMP Bulk Data entries) METHOD<sub>i</sub> = 1 option should be used. For element loading entries (like PLOAD, PLOAD2, PLOAD4, and TEMPP1 Bulk Data entries) either METHOD<sub>i</sub> = 0 or METHOD<sub>i</sub> = 1 option can be used. In particular, if harmonic output of element stresses under temperature loading via TEMPP1 Bulk Data entry, METHOD<sub>i</sub> = 0 option should be used to specify TEMPP1 load set.

**LOADOF**

Specifies Table IDs for Individual Degrees of Freedom for the Static Loads with Tables Described using LOADT Entries

Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.

Format:

| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|----|----|----|----|----|----|----|---|----|
| LOADOF | T0 | T1 | T2 | T3 | T4 | T5 | T6 |   |    |

Example:

|        |     |     |     |     |     |     |     |  |  |
|--------|-----|-----|-----|-----|-----|-----|-----|--|--|
| LOADOF | 100 | 101 | 102 | 103 | 104 | 105 | 106 |  |  |
|--------|-----|-----|-----|-----|-----|-----|-----|--|--|

| Descriptor    | Meaning   |
|---------------|---|
| T0            | Table ID specified as a negative Ti value on a LOADT entry (specified as a positive value on this entry. (Integer > 0; no Default)  |
| Ti (i=1 to 6) | ID of a TABLEL1 entry that specifies the time history of each degree of freedom for the loads specified by Li, -Ti on LOADT entries. Up to 6 degrees of freedom may be specified. See Remark 2. (Integer; no Default) |

Remarks:

1. LOADOF can only be used in SOL 600
2. This entry must not be used unless there is a negative value of Ti on one or more LOADT entries.

**LOADT**

Specifies Static Loads that use a Table to Describe Their Variation with Pseudo-Time

Specifies static loads that will use a table to describe their variation with pseudo-time. Available static entries are FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SPCD, SPCR, RFORCE and GRAV for SOL 600.

**Format:**

| 1     | 2   | 3  | 4  | 5    | 6  | 7  | 8  | 9  | 10 |
|-------|-----|----|----|------|----|----|----|----|----|
| LOADT | SID | L1 | T1 | L2   | T2 | L3 | T3 | L4 |    |
|       | T4  | L5 | T5 | etc. |    |    |    |    |    |

**Example:**

|       |     |   |    |   |    |   |    |  |  |
|-------|-----|---|----|---|----|---|----|--|--|
| LOADT | 101 | 1 | 51 | 2 | 52 | 3 | 53 |  |  |
|-------|-----|---|----|---|----|---|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number which must have a matching Bulk Data entry LOAD with the same SID. (Integer > 0; no Default)                  |
| Li         | Load set identification numbers defined on entry types listed above. (Integer > 0)   |
| Ti         | ID of a TABLEL1 or TABL3D0 entry that specifies the time history of the entry type. See Remarks 4. through 6. (Integer or Blank; no Default) |

**Remarks:**

1. LOADT can only be used in SOL 600.
2. In SOL 600 applied displacements must use SPCD (or SPCD) rather than SPC.
3. In SOL 600, if any LOADT entries are found in the model, the following parameters are automatically set:
  - PARAM,MARCVERR,11 (Allows table-driven inputs)
  - PARAM,MRCONVER,11 (Uses contact compatible with Version 11)
  - PARAM,MARCTOTT,1 (Allows loads with tables)
4. If Ti is positive, the table will be used for all degrees of freedom for the loads specified by Li. If Ti is negative, the absolute value of Ti will be used as the ID of a LOADOF entry providing the table IDs for each degree of freedom for which Li applies.
5. If more than one Li references the same force, moment pload4, etc, the first Ti encountered for that entity will be used for all occurrences of that entity for that subcase. Therefore, the 2nd and subsequent occurrences of Ti for these common entities may be left blank. An example can be found in ETL example mtloadt03.

6. It is strongly suggested that for a particular dof of a FORCE,MOMENT, etc. that only one table describe the variation for all subcases in the model. For example, if there are 3 subcases the time should normally cover the entire range of 0.0 to 3.0 (rather than having three tables, one for the range 0.0 to 1.0, a second for the range of 1.0 to 2.0 and a third from 2.0 to 3.0).

## LORENZI

## Fracture Mechanics J-Integral in SOL 600

This option gives an estimation of the J-Integral for a crack configuration using the domain integration method. The domain integration method has the advantage that it can also be used for problems with thermal behavior and for dynamic analysis. This procedure is only available for continuum elements. Only the nodes defining the crack front (crack tip in two dimensions) need to be defined. The program automatically finds integrations paths according to the format below. The complete J-Integral is evaluated and output. For the case of linear elastic material with no external loads on the crack faces, the program automatically separates mode I, mode II, and mode III (3-D only) stress intensity factors from the J-Integral.

Format:

| 1       | 2      | 3     | 4     | 5     | 6   | 7     | 8   | 9   | 10 |
|---------|--------|-------|-------|-------|-----|-------|-----|-----|----|
| LORENZI | IDCR   |       | MSEP  | ITYPE | G2D | ISET0 |     |     |    |
|         | DIRECT | ISETD | SV1   | SV2   | SV3 |       |     |     |    |
|         | TOP    | NREG  | JTYPE | TOL   | SV1 | SV2   | SV3 |     |    |
|         | GEOM   | NREG  | JTYPE | TOL   | SV1 | SV2   | SV3 | RAD |    |
|         |        | CYLIN |       |       |     |       |     |     |    |

Example:

|         |        |     |     |         |     |  |  |  |  |
|---------|--------|-----|-----|---------|-----|--|--|--|--|
| LORENZI | 1      |     | 0   | 1       | 1   |  |  |  |  |
|         | DIRECT | 101 | 0.0 | -2.0E-3 | 0.0 |  |  |  |  |
|         | DIRECT | 102 | 0.0 | -2.0E-3 | 0.0 |  |  |  |  |

|         |     |   |   |   |   |  |  |  |  |
|---------|-----|---|---|---|---|--|--|--|--|
| LORENZI | 1   |   | 0 | 1 | 1 |  |  |  |  |
|         | TOP | 2 | 1 |   |   |  |  |  |  |

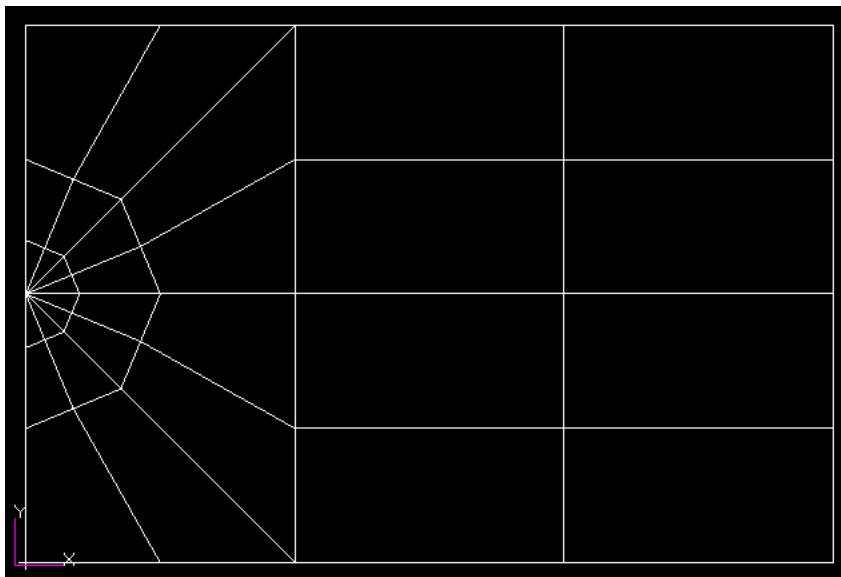
| Descriptor     | Meaning  |   |  |  |  |  |  |  |  |  |
|----------------|--|---|--|--|--|--|--|--|--|--|
| IDCR<br>(4-2)  | Identification of this particular crack. (Integer > 0; Required field) IDCR must be unique among all LORENZI cracks. |   |  |  |  |  |  |  |  |  |
| MSEP<br>(3-1)  | Enter 0 for no mode separation or 1 for mode separation. (Integer; Default = 1)                                      |   |  |  |  |  |  |  |  |  |
| ITYPE<br>(4-1) | Type of crack propagation (Integer; Default = 2)   |   |  |  |  |  |  |  |  |  |
|                | 1  | The “rigid region” will be described using direct inputs of nodes or elements.    |  |  |  |  |  |  |  |  |
|                | 2  | The “rigid region” will be described using an automatic search based on topology. |  |  |  |  |  |  |  |  |
|                | 3  | The “rigid region” will be described using an automatic search based on geometry. |  |  |  |  |  |  |  |  |

| Descriptor                    | Meaning  |
|-------------------------------|--|
| G2D<br>(5 for 2D)             | If the crack is 2D, enter the grid ID of the node at the crack tip. If the crack is 3D, leave this field blank. (Integer; no Default)  |
| ISET0<br>(5 for 3D)           | If the crack is 3D, enter the ID of a SET3 entry that defines an ordered set of grids or elements along the crack tip (for example in the thickness direction of the crack). If the crack is 2D, enter the grid ID of the node at the crack tip. (Integer; no Default) |
| DIRECT<br>(Start of 5a)       | Enter the string DIRECT if ITYPE=1, otherwise skip this line. See Remark 4. (Character,)   |
| ISETD<br>(6a)                 | Enter the ID of a SET3 entry defining an ordered set of grids or elements within the rigid region. See Remark 4. (Integer; no Default)   |
| SV1<br>(7a-1 or 6b-1 or 6c-1) | First component of shift vector. (Real; Default = .0.)   |
| SV2<br>(7a-2 or 6b-2 or 6c-2) | Second component of shift vector. (Real; Default = 0.0)  |
| SV3<br>(7a-3 or 6b-3 or 6c-3) | Third component of shift vector, enter only for 3D. (Real; Default = 0.0)  |
| TOP (Start of 5b)             | Enter the string TOP if ITYPE=2, otherwise skip this line. (Character)   |
| NREG<br>(5b-1 or 5c-1)        | Number of regions for which crack growth is to be estimated. See Remark 5. (Integer; Default = 1)  |
| JTYPE<br>(5b-2 or 6b-2)       | Enter 0 if SV1, SV2, SV3 on this line are to be used or enter 1 for automatic determination of SV1, SV2 and SV3. (Integer; Default = 1)  |
| TOL<br>(5b-3 or 5c-3)         | Enter tolerance for multiple grids at the crack tip. (Real; Default = 0.0)   |
| GEOM<br>(Start of 5c)         | Enter the string GEOM if ITYPE=3, otherwise skip this line (Character)   |
| RAD<br>(7c-1)                 | Radius of the rigid region to be found. (Real; no Default) Enter only if ITYPE=3.  |
| CYLIN<br>(7c-2)               | Relative length of cylinder for the path search. (Real; no Default) Enter only if ITYPE=3.   |

**Remarks:**

1. This entry corresponds to Marc's LORENZI model definition option.
2. (i,j) corresponds to Marc Vol C LORENZI entry ith datablock jth field.
3. Repeat the LORENZI as many times as necessary to define all cracks for which J-Integrals should be evaluated.

4. If the DIRECT input is used, repeat the DIRECT line as many times as necessary to define different regions from which crack growth should be estimated. In some cases it is informative to calculate crack growth for several regions and compare the results of these calculations. For example, in the model shown below, the crack tip is at the center on the left side. It might be informative to compare crack growth using a region including all nodes in the first ring (of node) around the crack with that produced by two rings.
5. For the TOP and GEOM methods, only one line per crack should be entered.
6. The continuation line for GEOM is not required except for 3D cracks.
7. For the previous examples, the crack tip is at grid 1 in the following figure. For the direct input, SET3 with ID 101 would reference all grids on and within the ring closest to the crack tip. SET3 with ID 102 would reference all grids on and within the two rings closest to the crack tip. Both examples provide identical results. The topology input is considerably simpler and therefore recommended.
8. The definition of shift vector is the function  $q_1$  in the equation and text described below.



## Background

The J-integral evaluation is based upon the domain integration method. A direct evaluation is not very practical in a finite element analysis due to the difficulties in defining the integration path. In the domain integration method for two dimensions, the line integral is converted into an area integration over the area inside the path. This conversion is exact for the linear elastic case and also for the nonlinear case if the loading is proportional, that is, if no unloading occurs. By choosing this area as a set of elements, the integration is straightforward using the finite element solution. In two dimensions, the converted expression is

$$\bar{J} = \int_A \left( \sigma_{ij} \frac{\partial u_j}{\partial x_1} - W \delta_{1i} \right) \frac{\delta q_1}{\delta x_i} dA$$

for the simplified case of no thermal strains, body forces or pressure on the crack faces. A is the area inside the integration path and  $q_1$  is a function introduced in the conversion into an area integral. The function can be chosen fairly generally, as long it is equal to one at the crack tip and zero on. The form of the function chosen is that it has the constant value of one at all nodes inside, and decreases to zero over the outermost ring of elements. It can be interpreted as a rigid translation of the nodes inside while the nodes on remain fixed. Thus, the contribution to the above equation comes only from the elements in a ring away from the crack tip. This interpretation is that of virtual crack extension and this method can be seen as a variant of such a technique, although it is extended with the effects of thermal strains, body forces, and pressure on the crack faces. The set of nodes moved rigidly is referred to as the *rigid region* and the function  $q_1$  in the above equation as the **shift vector**. For the evaluation of the J-integral the direction of the shift vector is simply the x axis in the local crack tip system.

In three dimensions, the line integral becomes an area integral where the area is surrounding a part of the crack front. In this case, the selection of the area is even more cumbersome than in two dimension. The converted integral becomes a volume integral which is evaluated over a set of elements. The rigid region is a set of nodes which contains a part of the crack front, and the contribution to the integral comes from the elements which have at least one but not all its nodes in the rigid region.

**LSEQ****Static Load Set Definition**

Defines a sequence of static load sets.

**Format:**

| 1    | 2   | 3        | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|------|-----|----------|-----|-----|---|---|---|---|----|
| LSEQ | SID | EXCITEID | LID | TID |   |   |   |   |    |

**Example:**

|      |     |     |      |      |  |  |  |  |  |
|------|-----|-----|------|------|--|--|--|--|--|
| LSEQ | 100 | 200 | 1000 | 1001 |  |  |  |  |  |
|------|-----|-----|------|------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification of the set of LSEQ entries. See Remark 5. (Integer > 0)  |
| EXCITEID   | The EXCITEID set identification assigned to this static load vector. See Remark 5. (Integer > 0)  |
| LID        | Load set identification number of a set of static load entries such as those referenced by the LOAD Case Control command. (Integer > 0 or blank)        |
| TID        | Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control command. (Integer > 0 or blank) |

**Remarks:**

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
3. EXCITEID may be referenced by CLOAD, RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries in order to apply the static load in nonlinear, static and dynamic analysis.
4. Element data recovery for thermal loads is not currently implemented in dynamics.
5. The SID-EXCITEID number pair must be unique with respect to similar pairs on all other LSEQ entries in the Bulk Data.
6. In a nonsuperelement analysis, LID and TID cannot both be blank. In superelement analysis, they may both be blank as long as static loads are prescribed in the upstream superelements.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LSEQ Bulk Data entries, all static loads whose load set IDs match the EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.

## Entries M - O

### MACREEP

### AUTO CREEP Iteration Control in SOL 600

Controls a transient creep analysis. This entry or the MTCREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600.

Format:

| 1       | 2   | 3    | 4    | 5    | 6   | 7   | 8     | 9   | 10 |
|---------|-----|------|------|------|-----|-----|-------|-----|----|
| MACREEP | ID  | Tinc | Ttot | Nmax | NIM | NIK | Tstab | VV1 |    |
|         | VV2 | VV3  | IABS |      |     |     |       |     |    |

Example:

|         |      |     |       |       |   |   |  |     |  |
|---------|------|-----|-------|-------|---|---|--|-----|--|
| MACREEP | 1    | 1.0 | 1000. | 99999 | 5 | 1 |  | .01 |  |
|         | 100. | .05 | 0     |       |   |   |  |     |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of a matching Case Control NLPARM command (for statics) or command (for dynamics). (Integer; Required Field) |
| Tinc       | Suggested time increment for creep analysis - The specified value will automatically be adjusted. (Real; Required Value) (2,1)     |
| Ttot       | Total time (final time) of the creep analysis. (Real; Required Value) (2,2)  |
| Nmax       | Maximum number of time increments allowed in the creep analysis (Integer; Default = 50) (2,3)                                      |
| NIM        | Maximum number of iterations allowed to modify a time step (Integer; Default = 5) (2,4)  |
| NIK        | Number of increments between stiffness matrix updates (Integer; Default = 1) (2,5)   |
| Tstab      | Stable time step, if known, must be entered for viscoplasticity (Real; Required field for viscoplasticity) (2,7)                   |
| VVT        | Tolerance value #1. (Real; see below for Defaults) (3,1)   |
|            | IABS=0 Enter the tolerance on the creep strain increment divided by the elastic strain (Default = 0.5).                            |
|            | IABS=1 Enter the maximum creep strain increment allowed. (Default = 0.1)   |
| VV2        | Tolerance value #2. (Real, see below for defaults) (3,2)   |
|            | IABS=0 Enter the tolerance on the stress change divide by the total stress. (Default = 0.1)  |
|            | IABS=1 Enter the maximum stress increment. (Default = 100.0)   |

| Descriptor | Meaning  |
|------------|--|
| VV3        | Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking (Real; Default = 0.05) (3,3) |
| IABS       | Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (3,5)  |
| 0          | Relative checking is used  |
| 1          | Absolute checking is used  |

**Remarks:**

1. This entry maps to Marc's AUTO CREEP entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file for creep analysis. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entry MPCREEP must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

**MARCIN****Inserts a Text String in Marc from SOL 600**

Inserts a text string directly in the Marc input file used in SOL 600 only.

**Format:**

| 1      | 2    | 3     | 4 | 5 | 6      | 7 | 8 | 9 | 10 |
|--------|------|-------|---|---|--------|---|---|---|----|
| MARCIN | ILOC | ICONT |   |   | String |   |   |   |    |

**Example:**

|        |    |   |              |  |
|--------|----|---|--------------|--|
| MARCIN | -1 | 0 | feature,2301 |  |
|--------|----|---|--------------|--|

| Descriptor | Meaning  |
|------------|--|
| ILOC       | Identification of the location in the Marc file where the string will be placed. (Required; Integer)   |
|            | -1 String is placed at end of Marc's Parameter Section   |
|            | 0 String is placed at the end of Marc's Model Definition Section   |
|            | N (N > 0) String is placed in load case N after the AUTO STEP, AUTO INCREMENT, etc. entry.   |
| ICONT      | (Required; Integer)  |
|            | -1 String is used as the name of an include file which contains all of the direct Marc input data for that portion of Marc (parameter, model definition, etc.). Only one include file per Marc Section is allowed. The string must be entirely in lower case and the include file must be in lower case, except for PC systems where the case does not matter. |
|            | 0 Entry is not a continuation of previous MARCIN entry.  |
|            | 1 Entry is a continuation of previous MARCIN entry and the strings will be placed one after the other on the same Marc line  |
| String     | (Required; Character)  |
|            | 0 The desired text string. The string is limited to 48 characters per entry. Multiple entries will be placed in the order entered within each Marc location.   |
|            | -1 String is an include file name limited to 48 characters.  |

**Remarks:**

1. Standard Nastran fields 4-9 are ignored for this entry. The string may be entered anywhere within fields 4-9 and will be translated directly to Marc.
2. If a long line for the Marc data is required, enter as many MARCIN entries as necessary to describe the entire Marc string using ICONT=1 for each except the first.

3. The total string length including continuation lines is limited to 160 characters.
4. As many MARCIN entries as necessary may be entered to define all desired input.
5. Each entry in the Nastran data file must start with the MARCIN header. Each line in an include file will be translated directly to Marc (there should be no MARCIN, ILOC or ICONT information in the include file(s)).
6. If the direct Marc input is placed in include file(s), separate files are necessary for each portion of Marc (parameter, model definition, etc.) requiring direct input.
7. As part of the Nastran input process, all strings are converted to uppercase. The internal Marc translator will convert them to lower case. For input entered without include files, this will normally make any difference. For include file names, file names must be entirely lower case for computer systems that are case sensitive.
8. MARCIN entries are not always recognized in restart runs and are not recommended.
9. ILOC greater than zero is not available for SOL 600 heat transfer analysis or structural analysis if table-driven loads are requested (for example, by specifying param,marctott,1).
10. For ILOC > 0 if STRING starts with "\$", "comment", or "COMMENT", the string will usually be skipped and no continuation lines should be entered.

## MARCOUT

Selects Data Recovery Output in SOL 600

Selects output to be saved on the Marc t16 end/or t19 file(s) used in SOL 600 only. This entry is available using the small field format only and should normally be used only when post-processing using the T16 file is to be done (in other words, it should not normally be used if OUTR on the SOL 600 entry is selected - see PARAM,MARROUTT).

Format:

| 1       | 2       | 3   | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|---------|-----|------|-----|-----|-----|-----|-----|----|
| MARCOUT | LAYCODE | IO1 | IO2  | IO3 | IO4 | IO5 | IO6 | IO7 |    |
|         | IO8     | IO9 | etc. |     |     |     |     |     |    |

Example:

|         |     |     |     |     |    |    |     |  |  |
|---------|-----|-----|-----|-----|----|----|-----|--|--|
| MARCOUT | 125 | E11 | E21 | E41 | N1 | N2 | N35 |  |  |
|---------|-----|-----|-----|-----|----|----|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| LAYCODE    | Specifies which shell or beam layers are to be output. (Integer; Default is all layers from 1 to the value of PARAM,MARCSLHT) This parameter also applies to composites whether made of shells or solids. For options less than 100, output will be all integration points. For options 100-200, output will only be at the element center. See Remark 13. |
| 0          | All shell and beam layer results from 1 to MARCSLHT are output, solid output is available only at the centroid. This option also requires PARAM,MROUTLAY to be set to the maximum numbers of layers desired (MARCSLHT and MROUTLAY should be set to the same values).  |
| 1          | Top and Bottom layer shell, beam and solid bottom layer results only are output (Marc layer codes 1 and 1000).   |
| 2          | Top, Bottom and Middle layer results for shell beams and centroid results for solids only are output (Marc layer codes 1, 5000 and 10000)  |
| 3          | Top and Bottom layer shell, beam and solid bottom layer results are output. In addition, composite layer output listed in the Bulk Data entry, MLAYOUT will be included.   |
| 100        | Same as option 0 except output is only at the center of each layer   |
| 101        | Same as option 1 except output is only at the center of each layer   |
| 102        | Same as option 2 except output is only at the center of each layer   |
| 103        | Same as option 3 except output is only at the center of each layer.  |
| IO(i)      | Indicates the type of Marc output requested according to the following table:  |

| Element Output | Description   |
|----------------|---|
| -9999          | If -9999 is entered in field 3, the defaults for the post file of the Marc version being run will be used. Fields 4-9 must be blank and no continuation lines should be entered. No OUTR options on the SOL 600 entry are available with this option. |
| "E-USER"*      | First user-defined element post code(s) are generated by user subroutine plotv.f  |
| "E-USER1"*     | 2 <sup>nd</sup> user-defined element post code(s)   |
| "E-USER2"*     | 3 <sup>rd</sup> user-defined element post code(s)   |
| "E-USER3"*     | 4 <sup>th</sup> user-defined element post code(s)   |
| "E-USER99"*    | 100 <sup>th</sup> user-defined element post code(s)   |

\*These outputs are only available in the t16 file, not in op2, xdb, f06, punch.

A maximum number of 100 user-defined element post codes may be entered for SOL 600).

|            |   |
|------------|---|
| E1 to E7   | strain components, VM strain                                |
| E8         | Equivalent plastic strain                                   |
| E9         | Total Temperature (combined heat/structural analysis)       |
| E10        | Incremental Temperature (combined heat/structural analysis) |
| E11 to E17 | stress components, VM stress, Mean stress                   |
| E18        | Mean normal stress  |
| E20        | Element Thickness (thickness can change vs time)            |
| E21 - E27  | plastic strain components, VM plastic strain                |
| E28        | Plastic strain rate   |
| E29        | Value of second state variable if applicable                |
| E31 to E37 | Creep strain components and equivalent creep strain         |
| E38        | Total swelling strain                                       |
| E39        | Value of third state variable if applicable                 |
| E41 to E47 | Cauchy stress components, VM Cauchy stress                  |
| E49        | Thickness strain for plane stress, Mooney or Ogden material |
| E58        | Elastic strain energy density                               |
| E59        | Equivalent stress/yield stress                              |
| E60        | Equivalent stress/yield stress at current temperatures      |
| E48        | Strain Energy Density                                       |
| E68        | Plastic strain energy density                               |
| E69        | Current Volume  |
| E71 to E76 | Components of thermal strain (separated from total strain)  |
| E78        | Original Volume   |

| Element Output | Description  |
|----------------|--|
| E79            | Grain size if applicable   |
| E80            | Damage indicator for Cockcroft-Latham, Oyane and principal stress criteria |
| E81 to E86     | Components of cracking strain if applicable                                |
| E91 to E107    | Failure indices associated with failure criteria from MATF                 |
| E108 to E109   | Interlaminar shear for thick composite shells                              |
| E110           | Interlaminar shear bond index for thick composite shells                   |
| E111 to E116   | Components of stress in "preferred coordinate system"                      |
| E121 to E126   | Elastic strain components  |
| E127           | Equivalent elastic strain  |
| E128           | Major engineering strain   |
| E129           | Minor engineering strain   |
| E171           | Porosity   |
| E172           | Void ratio   |
| E173           | Pore pressure  |
| E174           | Preconsolidation pressure  |
| E175           | Equivalent viscoplastic strain rate (powder material)                      |
| E176           | Relative density (powder material)   |
| E177           | Void volume fraction (damage model)  |
| E178           | Lemaitre damage factor   |
| E179           | Lemaitre relative damage   |
| E180           | Total temperature (same as E9)   |
| E181 to E183   | Components of temperature gradient   |
| E184 to E186   | Components of flux   |
| E241           | Gasket pressure if applicable  |
| E242           | Gasket closure if applicable   |
| E243           | Plastic Gasket closure if applicable                                       |
| E244           | Exponential powder parameter kappa   |
| E245           | Exponential powder parameter x   |
| E251 to E253   | Components of interlaminar normal stress in basic coordinate system        |
| E254 to E256   | Components of interlaminar shear stress in basic coordinate system         |
| E257           | Interlaminar shear bond index for composite solids                         |
| E216           | Beam axis or Orientation of CBUSH/CFAST elements                           |
| E261           | Beam Axis  |

| Element Output | Description  |
|----------------|--|
| E264-E269      | Beam/Bar or Bush Element Forces  |
|                | The next 6 lines are for CBUSH Force output (CBUSH stress/strain output and can be obtained with standard stress/strain codes if the stress/strain recovery. Coefficients are defined on the PBUSH entries.) |
| E264           | CBUSH Axial Force (local bush coord sys)   |
| E265           | CBUSH Moment Mxx (local bush coord sys)  |
| E266           | CBUSH Moment Myy (local bush coord sys)  |
| E267           | CBUSH Shear Force Vxy (local bush coord sys)   |
| E268           | CBUSH Shear Force Vyx (local bush coord sys)   |
| E269           | CBUSH Torque (local bush coord sys)  |
| E270           | Bimoment (for beam element)  |
| E301           | Total strains tensor   |
| E311           | Stress tensor  |
| E321           | Plastic strain tensor  |
| E331           | Creep strain tensor  |
| E341           | Cauchy stress tensor   |
| E371           | Thermal strain tensor  |
| E381           | Cracking strain tensor   |
| E391           | Stress in composite ply ("preferred") direction  |
| E401           | Elastic strain tensor  |
| E411           | Stress tensor in basic coordinate system   |
| E421           | Elastic strain tensor in basic coordinate system   |
| E431           | Plastic strain tensor in basic coordinate system   |
| E441           | Creep strain tensor in basic coordinate system   |
| E451           | Velocity strains (for fluids)  |
| E461           | Elastic strain in preferred direction tensor   |
| E471           | Rebar stress components in basic coordinate system in undeformed config if applicable  |
| E481           | Rebar stress components in basic coordinate system in deformed config if applicable  |
| E487           | Rebar angle  |
| E501           | Interlaminar normal stress   |
| E511           | Interlaminar shear stress  |
| E531           | Volume fraction of Martensite  |

| Element Output | Description  |
|----------------|--|
| E541           | Phase transformation strain tensor                             |
| E547           | Equivalent phase transformation strain                         |
| E548           | Equivalent TWIN strain   |
| E549           | Equivalent TRIP strain   |
| E551           | Equivalent plastic strain in a multiphase aggregate            |
| E552           | Equivalent plastic strain in Austenite                         |
| E553           | Equivalent plastic strain in Martensite                        |
| E557           | Yield stress of multiphase aggregate                           |
| E610 to E617   | Strength ratios based on MATF failure modes                    |
| E641           | Generalized strain curvatures tensor                           |
| E661           | Generalized stress moments tensor                              |
| E681           | True strain tensor for continuum elements                      |
| E691           | Element orientation vector 1                                   |
| E694           | Element orientation vector 2                                   |
| E697           | Layer orientation angle  |
| E704           | Real harmonic axial force                                      |
| E705           | Real harmonic Moment Mxx                                       |
| E706           | Real harmonic Moment Myy                                       |
| E707           | Real harmonic Shear force Vxy                                  |
| E708           | Real harmonic Shear force Vy়                                  |
| E709           | Real harmonic Torque   |
| E710           | Real harmonic Bimoment   |
| E714           | Imaginary harmonic axial force                                 |
| E715           | Imaginary harmonic Moment Mxx                                  |
| E716           | Imaginary harmonic Moment Myy                                  |
| E717           | Imaginary harmonic Shear force Vxy                             |
| E718           | Imaginary harmonic Shear force Vy়                             |
| E719           | Imaginary harmonic Torque                                      |
| E720           | Imaginary harmonic Bimoment                                    |
| E721           | Cauchy Stress Tensor in Preferred (Material) Coordinate System |
| E731           | Curvature Tensor (for shell elements)                          |
| E741           | Moment Tensor (for shell elements)                             |

| Nodal Output  | Description   |
|---|---|
| "N-USER"*   | First user-defined nodal post cod are generated by user subroutine upstnd.f                   |
| "N-USER1"*  | 2 <sup>nd</sup> user-defined nodal post cod are generated by user subroutine upstnd.f         |
| "N-USER2"*  | 3 <sup>rd</sup> user-defined nodal post cod are generated by user subroutine upstnd.f         |
| "N-USER3"*  | 4 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f         |
| "N-USER4"*  | 5 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f         |
| "N-USER99"*   | 100 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f, etc. |
| *User-defined outputs are only available in the t16 file, not in op2, xdb, f06, punch. A maximum of 100 user-defined nodal post codes may be entered for SOL 600. |   |
| N1,N2   | displacements, rotations  |
| N3,N4   | Applied forces & moments  |
| N5,N6   | reaction forces & moments   |
| N7  | Fluid velocity if applicable  |
| N8  | Fluid pressure if applicable  |
| N9  | External fluid force if applicable  |
| N10   | Reaction fluid force if applicable  |
| N11   | Sound pressure if applicable  |
| N12   | External sound pressure if applicable   |
| N13   | Reaction sound pressure if applicable   |
| N14   | Temperature   |
| N15   | External heat flux  |
| N16   | Reaction heat flux  |
| N23   | Pore pressure if applicable (soil analysis)   |
| N24   | External mass flux if applicable  |
| N25   | Reaction mass flux if applicable  |
| N26   | Bearing pressure if applicable  |
| N27   | Bearing force if applicable   |
| N28,N29   | Velocity  |
| N30,N31   | Acceleration  |
| N32,N33   | Modal mass and rotational modal mass  |
| N34,N35   | Contact normal stress/force   |
| N36,N37   | Contact friction stress/force   |
| N38,N39   | Contact status, Contact touched body  |

| Nodal Output | Description                     |
|--------------|---------------------------------|
| N40          | Herrmann variable               |
| N46,N48      | Tying force and moments         |
| N47          | Coulomb Force                   |
| N49          | Generalized nodal stress        |
| N50          | Generalized nodal strain        |
| N51,N52      | Inertia relief force and moment |
| N53          | J-Integral                      |
| N54          | Stress Intensity, Mode I        |
| N55          | Stress Intensity, Mode II       |
| N56          | Stress Intensity, Mode III      |
| N57          | Energy release                  |
| N58          | Energy release rate I           |
| N59          | Energy release rate II          |
| N60          | Energy release rate III         |
| N62          | Crack system local X            |
| N63          | Crack system local Y            |
| N64          | Crack system local Z            |
| N65          | Contact Separation Distance     |
| N66          | Normal Breaking Index           |
| N67          | Tangential Breaking Index       |
| N68          | Total Breaking Index            |
| N69          | Normal Delamination Index       |
| N70          | Tangential Delamination Index   |
| N71          | Total Delamination Index        |
| N73          | Glue deactivation status        |
| N74          | VCCT Failure Index              |
| N76          | Lorentz Force                   |
| N77          | Wear Index                      |
| N78          | Wear Rate                       |

**Remarks:**

1. MARCOUT is only available when Marc is executed from within Nastran and controls what results are available in the Marc t16 file. All elements or nodes of each type selected will be placed on the t16 file (in other words, it is not possible to control this output by selecting various sets). Some, but not all, of the results in the t16 file may be used to obtain .op2, .xdb, punch or .f06 results output by specifying OUTR options on the SOL 600 Executive Control statement. The type of results which may be placed in the .op2, .xdb, punch or .f06 files are the same as those available in SOL 106 or 129. Other types of results are only available for postprocessing using the t16 file. Whenever possible, i.e., if your gui allows it, the t16 file should be used instead of .op2, .xdb, punch, or .f06 files for postprocessing.
2. Values such as E1, E21 correspond to Marc's postcodes 1 and 21, respectively.
3. Outputs produced by MARCOUT are the same for all subcases, load steps, iterations, etc.
4. The MARCOUT entry may be repeated as many times as desired, or all entries may be placed on continuation lines.
5. For entries E1, E11 and E21 corresponding entries E2-E7, E12-E17 and E22-E27 will be generated automatically. These terms correspond to 3 normal stress (or strain) and 3 shear stress/strain values plus the equivalent von Mises value. See Marc volume C POST description for more details.
6. If this entry is not used, the following defaults are entered automatically: E301, E341, E47, N1, N2, N35, N37, N39. When creep or heat transfer is present, additional items are added appropriately. If the MARCOUT entry is entered, only those items specified will be output.
7. Only displacements, rotations, Cauchy stresses and one type of strains (total, plastic or elastic) may be transferred to the Nastran database.
8. Displacements, at least one stress tensor and one strain tensor must be selected if any OUTR options are to be used.
9. For SOL 600, Nastran Case Control commands such as SET ID=, DISP=, STRESS=, and STRAIN= only control the output in the .OP2, .XDB, punch, .F06 and jid.marc.out files. The Case Control requests do not affect the t16 output.
10. Default MARCOUT options are sufficient for most needs and it is recommended that the MARCOUT entry only be employed by advanced users.
11. If some (but not all) forces (E264-E269) are specified, the missing ones will be added automatically since the t16op2 conversion requires all be present (this capability is available starting in MD Nastran R2 and the MSC Nastran 2006 release, prior to that, all needed to be specified if any OUTR options were requested.)
12. LAYCODE values 100-103 will automatically set PARAM,MARCCENT,1
13. For LAYCODE=1, if op2.xdb,f06 or punch output is requested element stress/strain results which normally are output at the center of the element will be output on the bottom surface of the element. For composite solids, if output at all layers is desired, postprocessing using the t16 file is required.
14. Additional outputs described in Marc Vol C (POST Section) may be available for certain types of SOL 600 analyses.

**MARPRN****Defines "Print" Options for SOL 600**

This option corresponds to Marc's PRINT parameter which controls a variety of output and other information. for the most used options, PARAM,MARCPRN should be used.

**Format:**

| 1      | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| MARPRN | IP1 | IP2    | IP3 | IP4 | IP5 | IP6 | IP7 | IP8 |    |
|        | IP9 | -etc.- |     |     |     |     |     |     |    |

**Example:**

|        |   |   |    |  |  |  |  |  |  |
|--------|---|---|----|--|--|--|--|--|--|
| MARPRN | 2 | 5 | 39 |  |  |  |  |  |  |
|--------|---|---|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IPi        | "Print" code from list shown below (Integer; no Default). Include as many options as desired   |
| 1          | Output element stiffness matrices (this also prints out the shell surface metric for doubly curved shells 4, 8, and 24), consistent mass matrix, and equivalent nodal loads.<br><br><b>CAUTION:</b> This produces significant output.  |
| 2          | Output of the matrices used in tying. (TYING, SERVO LINK, UFORMS.)   |
| 3          | Forces the solution of a nonpositive definite matrix. This is only recommended for the AUTO INCREMENT option to pass collapse points in the collapse analysis. This can be entered on the CONTROL option.                              |
| 5          | To obtain additional information concerning gap convergence. In contact analysis, set to 5 to obtain information concerning nodes touching or separating from surfaces and also to print out the maximum residual and reaction forces. |
| 6          | To obtain output of nodal value array during rezoning.   |
| 8          | To obtain incremental displacements in local system in contact problems.   |
| 9          | To obtain latent heat output.  |
| 10         | To obtain the stress-strain relation in the local coordinate system.   |
| 11         | To obtain additional information on the interlaminar stress calculation.   |
| 12         | To output the right-hand side and solution vector. <b>CAUTION:</b> This produces significant output.   |
| 13         | To obtain additional information regarding CPU resources used.   |
| 15         | To obtain additional information regarding surface energy balances.  |
| 20         | To obtain information regarding the evaluation of tables.  |

| Descriptor | Meaning  |
|------------|--|
| 21         | To obtain information about application of kinematic boundary conditions when table input is used.                               |
| 22         | To obtain information about distributed loads, point loads, films, foundations, and initial conditions when table input is used. |
| 26         | To print additional information regarding sink points.   |
| 27         | To obtain reaction forces at tied nodes.   |
| 28         | To obtain additional information about convective terms in heat transfer and fluid analysis.                                     |
| 34         | To print a description of what independent variables may be used with a physical quantity.                                       |
| 36         | To obtain CASI solver debug information (has the least details).   |
| 37         | To obtain CASI solver debug information (has more details).  |
| 38         | To obtain CASI solver debug information (has the most details).  |
| 39         | To obtain detailed information about memory allocation.  |
| 43         | To obtain information about VCCT   |
| 44         | To obtain information during progressive failure calculations  |

Remark:

1. See PARAM,MARCPRN for a simpler way to enter the most used print options.

**MASSET****Mass Combination Definition**

The MASSSET bulk data entry defines a linear combination of mass cases to form the subcase-dependent mass which is selected by associated MASSSET case control. The subcase mass is defined as follows by the MASSSET bulk:

$$M_{ID} = S_0 \sum_i S_i M_i$$

Where,  $i$  refers to the MASSID qualifier:  $i = 0$  is the base mass case,  $i > 0$  are the incremental mass cases. ID's of mass combinations defined by MASSSET bulk data entry and mass increments defined by MASSID bulk data sections should be unique.

Format:

| 1      | 2  | 3   | 4  | 5   | 6  | 7   | 8  | 9   | 10 |
|--------|----|-----|----|-----|----|-----|----|-----|----|
| MASSET | ID | S0  | S1 | ID1 | S2 | ID2 | S3 | ID3 |    |
|        | S4 | ID4 |    |     |    |     |    |     |    |

Example:

|        |    |     |     |   |     |     |  |  |  |
|--------|----|-----|-----|---|-----|-----|--|--|--|
| MASSET | 11 | 1.0 | 1.0 | 0 | 1.0 | 101 |  |  |  |
|--------|----|-----|-----|---|-----|-----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Set identification number referenced by MASSSET case control. (Integer > 0; Required) |
| S0         | The overall scale factor for the linear combination (Real >= 0.0; Default = 0.0)      |
| Si         | The scale factor for the ith mass increment (Real >= 0.0; Default = 0.0)              |
| IDi        | The MASSID for the ith mass increment (Integer > 0; No Default)                       |

**MATBV****Material Bulk Viscosity - SOL700**

Defines the bulk viscosity for materials. Used in SOL700 only.

**Format:**

| 1     | 2   | 3       | 4     | 5     | 6      | 7      | 8       | 9 | 10 |
|-------|-----|---------|-------|-------|--------|--------|---------|---|----|
| MATBV | MID | BULKTYP | BULKQ | BULKL | CSCALE | DMPFAC | IIMMREL |   |    |

**Example:**

|       |   |     |  |  |  |  |  |  |  |
|-------|---|-----|--|--|--|--|--|--|--|
| MATBV | 7 | 1.2 |  |  |  |  |  |  |  |
|-------|---|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification. See Remark 1. (Integer > 0, Required.)  |
| BULKTYP    | Bulk viscosity type. (Character: default=DYNA)<br>DYNA            Standard DYNA3D model.<br>DYTRAN        Enhanced DYNA model.   |
| BULKL      | Linear bulk-viscosity coefficient. (Real $\geq$ 0.0; default=1.0)  |
| BULKQ      | Quadratic bulk-viscosity coefficient. (Real $\geq$ 0.0; default=0.0)   |
| CSCALE     | When this material model is used with MEMB shell elements (See PSHELL1), the compressive stresses in the principal directions will be scaled with this factor. CSCALE=0.0 results in a tension only material. See Remark 2. (Real $\geq$ 0.0; default=1.0) |
| DMPFAC     | When this material model is used with MEMB shell elements, damping is applied to the stresses. DMPFAC = 0.05 results in 5% damping applied to membrane stresses. See Remark 2. (Real $\geq$ 0.0; default=0.0)  |
| IIMMREL    | Relaxation factor used with the Initial Metric Method. This option is only used when this material model is used with MEMB shell elements and the IMM method is activated. See Remark 3. (0.0 < Real < 1.0; default=1.0e-3)                                |

**Remarks:**

1. The material number must refer to a basic material definition such as MAT1/MAT2/MAT8/MATORT/MATHE.
2. For air bag modeling the following values of CSCALE and DMPFAC are suggested:  
 $CSCALE = 0.1$   
 $DMPFAC = 0.05 \text{ to } 0.20$
3. The Initial Metric Method relaxation factor is used to slow down the expansion of the membrane elements during the calculation. The default is sufficient in most simulations.

**MAT1****Isotropic Material Property Definition**

Defines the material properties for linear isotropic materials.

**Format:**

| 1    | 2   | 3  | 4  | 5     | 6   | 7 | 8    | 9  | 10 |
|------|-----|----|----|-------|-----|---|------|----|----|
| MAT1 | MID | E  | G  | NU    | RHO | A | TREF | GE |    |
|      | ST  | SC | SS | MCSID |     |   |      |    |    |

**Example:**

|      |       |       |       |      |      |       |        |      |  |
|------|-------|-------|-------|------|------|-------|--------|------|--|
| MAT1 | 17    | 3.+7  |       | 0.33 | 4.28 | 6.5-6 | 5.37+2 | 0.23 |  |
|      | 20.+4 | 15.+4 | 12.+4 | 1003 |      |       |        |      |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material identification number. (Integer > 0)   |
| E          | Young's modulus. (Real $\geq 0.0$ or blank)   |
| G          | Shear modulus. (Real $\geq 0.0$ or blank)   |
| NU         | Poisson's ratio. (-1.0 < Real $\leq 0.5$ or blank)  |
| RHO        | Mass density. See Remark 5. (Real)  |
| A          | Thermal expansion coefficient. (Real)   |
| TREF       | Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 9. and 10. (Real; Default = 0.0 if A is specified.)   |
| GE         | Structural element damping coefficient. See Remarks 8., 9., and 4. (Real)   |
| ST, SC, SS | Stress limits for tension, compression, and shear are optionally supplied, used only to compute margins of safety in certain elements; and have no effect on the computational procedures. (Real $\geq 0.0$ or blank). See Remark 14. |
| MCSID      | Material coordinate system identification number. Used only for PARAM,CURV processing. See Parameters, 783. (Integer $\geq 0$ or blank)   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. The following rules apply when E, G, or NU are blank:
  - E and G may not both be blank.
  - If NU and E, or NU and G, are both blank, then both are set to 0.0.

- If only one E, G, or NU is blank, then it will be computed from the equation:  

$$E = 2 \cdot (1 + NU) \cdot G$$
. If this is not desired, then the MAT2 entry is recommended. If E, G, or NU are made temperature dependent by the MATT1 entry, then the equation is applied to the initial values only. Caution: Inconsistent E and G can result in NU >.5 which for three dimensional elasticity results in negative stress for positive strain. This can cause large max ratios.
3. If values are specified for all of the properties E, G, and NU, then it is recommended that the following relationship be satisfied:

$$\left| 1 - \frac{E}{2 \cdot (1 + NU) \cdot G} \right| < 0.01$$

If this relationship is not desired, then the MAT2 entry is recommended.

It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements as indicated in [Table 22](#). Therefore, it is recommended that only the applicable properties be specified for a given element.

**Table 22** Material Property Usage Versus Element Types

| Element Entry                       | E   | NU                 | G                        |
|-------------------------------------|---|--------------------|--------------------------|
| CROD<br>CBEAM<br>CBAR               | Extension and Bending                           | Not Used           | Torsion Transverse Shear |
| CQUADi<br>CTRIAi<br>CCONEAX         | Membrane, including In-plane Shear, and Bending |                    | Transverse Shear         |
| CSHEAR                              | Not Used  |                    | Shear                    |
| CRAC2D                              | All Terms                                       |                    | Not Used                 |
| CHEXA<br>CPENTA<br>CTETRA<br>CRAC3D | All Terms                                       |                    | Not Used                 |
| CTRIAX6                             | Radial, Axial, Circumferential                  | All Coupled Ratios | Shear                    |

- MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
- The mass density RHO will be used to compute mass for all structural elements automatically.
- Weight density may be used in field 6 if the value 1/g is entered on the PARAM,WTMASS entry, where g is the acceleration of gravity (see [Parameters, 783](#)).
- MCSID must be nonzero if PARAM,CURV is specified to calculate stresses or strains at grid points on plate and shell elements only.

8. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$ , by 2.0.
9. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP/PCOMPG entry.
10. TREF is used in two different ways:
  - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
  - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection.

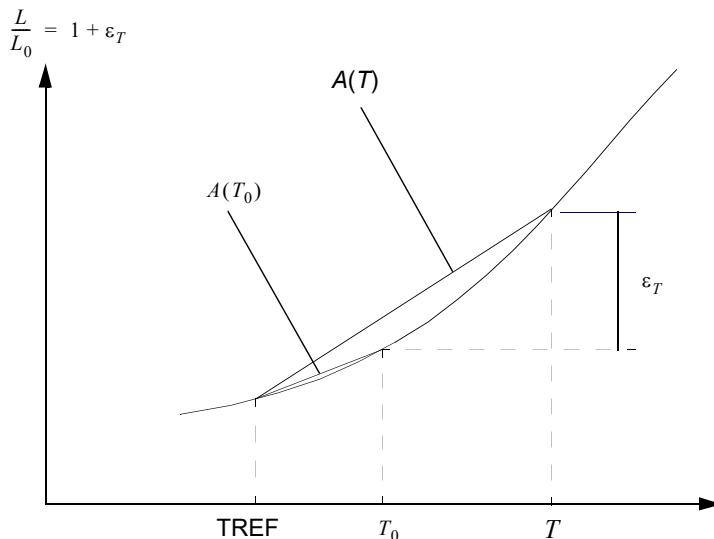


Figure 9-105 Use of TREF in Calculation of Thermal Loads

$$\varepsilon_T = A(T) \cdot (T - TREF) - A(T_0) \cdot (T_0 - TREF)$$

where  $T$  is requested by the TEMPERATURE(LOAD) command and  $T_0$  is requested by the TEMPERATURE(INITIAL) command.

**Notes:**

- A is a secant quantity.
- TREF is obtained from the same source as the other material properties; e.g., ASTM, etc.
- If  $A(T)$  constant, then  $\varepsilon_T = A \cdot (T - T_0)$
- If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).

11. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPISILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
12. For SOL 600, E must not be blank or zero.
13. Negative values for ST, SC, and SS lead to no margins of safety being computed.
14. ST, SC, and SS are not used in SOL 400 with advanced elements or SOL 600. Use MATF instead.

**MAT1A****Additional Material Properties for MAT1- SOL700**

Specifies additional the material properties of MAT1 for solid elements. Use SOL700 only.

Format:

| 1     | 2   | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|---|---|---|---|---|---|----|
| MAT1A | MID | EID |   |   |   |   |   |   |    |

Example:

|       |     |     |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|
| MAT1A | 101 | 201 |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MAT1. (Integer > 0; required.) |
| EID        | EOSxxx ID. (Integer > 0; required.)                     |

Remarks:

1. MAT1A always required for solid elements when MATEP is used. However, when RYIELD option of MATEP is set to VMISES, ORTHOCR or SOIL, MATEOS is not required.

**MAT1F****Isotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT1 entry fields via TABLEDi entries.

**Format:**

| 1     | 2   | 3 | 4 | 5  | 6   | 7 | 8 | 9  | 10 |
|-------|-----|---|---|----|-----|---|---|----|----|
| MAT1F | MID | E | G | NU | RHO |   |   | GE |    |

**Example:**

|       |    |    |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|
| MAT1F | 33 | 15 |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)   |
| E          | Identification number of a TABLEDi entry for the Young's modulus. (Integer > 0 or blank)  |
| G          | Identification number of a TABLEDi entry for the shear modulus. (Integer > 0 or blank)  |
| NU         | Identification number of a TABLEDi entry for the Poisson's ratio. (Integer > 0 or blank)  |
| GE         | Identification number of a TABLEDi entry for the structural damping coefficient. (Integer > 0 or blank)   |
| RHO        | Identification number of a TABLEDi entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only. |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT1 entry.
- The user is reminded that on the MAT1 entry, if any one of the entries E, G, or, NU is left blank, they are connected by the relationship  $G=E/(2(1+NU))$ . Therefore, this entry requires that if any E, G, or NU is to be frequency dependent then Table references must be present for all three of the E, G, NU even if this requires a table of constant values.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT1 then GE table must be blank or 0.

**MAT2****Shell Element Anisotropic Material Property Definition**

Defines the material properties for linear anisotropic materials for two-dimensional elements.

**Format:**

| 1    | 2     | 3    | 4    | 5    | 6    | 7    | 8    | 9   | 10 |
|------|-------|------|------|------|------|------|------|-----|----|
| MAT2 | MID   | G11  | G12  | G13  | G22  | G23  | G33  | RHO |    |
|      | A1    | A2   | A3   | TREF | GE   | ST   | SC   | SS  |    |
|      | MCSID | GE11 | GE12 | GE13 | GE22 | GE23 | GE33 |     |    |

**Example:**

|      |       |       |  |        |       |       |       |  |  |
|------|-------|-------|--|--------|-------|-------|-------|--|--|
| MAT2 | 13    | 6.2+3 |  | 6.2+3  |       | 5.1+3 | 0.056 |  |  |
|      | 6.5-6 | 6.5-6 |  | -500.0 | 0.002 | 20.+5 |       |  |  |
|      | 1003  |       |  |        |       |       |       |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification number. See Remark 1. (Integer > 0)  |
| Gij        | The material property matrix. (Real) See Remarks 4., 5., 6., and 9.  |
| RHO        | Mass density. (Real)   |
| Ai         | Thermal expansion coefficient vector. (Real)   |
| TREF       | Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 10., 11. and 14. (Real or blank)   |
| GE         | Structural element damping coefficient. See Remarks 8., 10., and 13. (Real)  |
| ST, SC, SS | Stress limits for tension, compression, and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. (Real or blank). See Remark 17. |
| MCSID      | Material coordinate system identification number. Used only for PARAM, CURV processing. See Parameters, 783. (Integer $\geq$ 0 or blank)   |
| GEij       | Structural damping matrix. See Remark 17. (Real)   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. MAT2 materials may be made temperature dependent by use of the MATT2 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density, RHO, will be used to automatically compute mass for all structural elements.

4. The convention for the Gij in fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A1 \\ A2 \\ A3 \end{Bmatrix}$$

The G matrix, including G13 and G23 is an extension to conventional orthotropic plans stress behavior that allows membrane-shear coupling. If the MAT2 is used for plane stress elements or referenced as MID1 or MID2 in a PSHELL, the G matrix must be invertible and positive definite.

5. When MAT2 is referenced as a MID3 in a PSHELL, then it is used as:

$$\begin{aligned} \tau_{13} &= \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{13} \\ \gamma_{23} \end{Bmatrix} \\ \tau_{23} & \end{aligned}$$

This may lead to user warning message 6134 which may be ignored. See *The NASTRAN Theoretical Manual*, Section 4.2.

6. When MAT2 is referenced as a MID4 in a PSHELL, then only G<sub>13</sub> and G<sub>23</sub> are required. One will observe a user warning message 9994, The Material Property Matrix is not positive definite which may be ignored.
7. MCSID must be nonzero if PARAM,CURV is specified to extrapolate element centroid stresses or strains to grid points on plate and shell elements only. CQUAD4 element corner stresses are not supported by PARAM,CURV.
8. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
9. Unlike the MAT1 entry, data from the MAT2 entry is used directly without adjustment of equivalent E, G, or NU values.
10. TREF and GE are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
11. TREF is used in two different ways:
  - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10. in the MAT1 description.
  - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
12. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
13. PCOMP/PCOMPG entries generate MAT2 entries starting from 10,000,001 onwards. Explicitly specified MAT2 IDs must not conflict with internally generated MAT2 IDs. Furthermore, if MID is greater than 400,000,000 then A1, A2, and A3 are a special format. They are [G4] · [α4] not [α4]. If MIDs larger than 99999999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.

14. In nonlinear static analysis (SOL 106) the QUAD4, TRIA3, QUADR, and TRIAR thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPISILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
15. Negative values for ST, SC, and SS lead to no margins of safety being computed.
16. If the MAT2 is referenced by the PCOMP/PCOMPG entry, the transverse shear flexibility for the referenced lamina is zero.
17. Defines structural damping matrix data for advanced composites. If the GE<sub>ij</sub> values are present, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GE<sub>ij</sub> values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDI entries must refer to MAT2 entries.

For any ply that does not point to a MAT2 with extended GE<sub>ij</sub>, the GE value will be taken from the GE field of the corresponding MAT<sub>i</sub> entry (MAT1 field 9, MAT2 field 6 of first continuation, MAT8 field 2 of second continuation.).

The damping when extended GE<sub>ij</sub> is in effect is applied in element material routines by forming the product GE<sub>ij</sub> \* G<sub>ij</sub> when forming the K4 damping matrix. For any ply not using an extended GE<sub>ij</sub> of MAT2 the product GE\*G<sub>ij</sub> is used.

**MAT2F****Shell Element Anisotropic Frequency Dependence**

Specifies frequency-dependent material properties on MAT2 entry fields via TABLEDi entries.

**Format:**

| 1     | 2   | 3    | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|-------|-----|------|------|------|------|------|------|---|----|
| MAT2F | MID | G11  | G12  | G13  | G22  | G23  | G33  |   |    |
|       |     |      |      |      | GE   |      |      |   |    |
|       |     | GE11 | GE12 | GE13 | GE22 | GE23 | GE33 |   |    |

**Example:**

|       |    |    |    |    |    |    |    |  |  |
|-------|----|----|----|----|----|----|----|--|--|
| MAT2F | 34 |    |    |    |    |    |    |  |  |
|       |    |    |    |    |    |    |    |  |  |
|       |    | 47 | 48 | 51 | 47 | 48 | 51 |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on MAT2 entry. (Integer > 0)      |
| Gij        | Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)                |
| GE         | Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)  |
| GEij       | Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank) |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT2 entry.
- Table references must be present for each item that is frequency dependent.
- If the GEij values are present on the MAT2 entry for any ply, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GEij values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix

which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDI entries must refer to MAT2 entries. See Remarks 17. and of MAT2 entry. The damping is applied in element material routines by forming the product  $GE_{ij} * G_{ij}$  when forming the K4 damping matrix.

4. IF  $GE=0.0$  on corresponding MAT2 then GE table must be blank or 0. For any  $GE_{ij}=0.0$  on corresponding MAT2 the corresponding  $GE_{ij}$  table must be blank or 0.

**MAT3**

## CTRIAX6 or PSHLN2 or PLCOMP Material Property Definition

Defines the material properties for linear orthotropic materials used by the CTRIA6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries.

**Format:**

| 1    | 2   | 3  | 4   | 5  | 6     | 7     | 8    | 9   | 10 |
|------|-----|----|-----|----|-------|-------|------|-----|----|
| MAT3 | MID | EX | ETH | EZ | NUXTH | NUTHZ | NUZX | RHO |    |
|      |     |    | GZX | AX | ATH   | AZ    | TREF | GE  |    |

**Example:**

|      |    |       |       |       |       |       |      |       |  |
|------|----|-------|-------|-------|-------|-------|------|-------|--|
| MAT3 | 23 | 1.0+7 | 1.1+7 | 1.2+7 | .3    | .25   | .27  | 1.0-5 |  |
|      |    |       | 2.5+6 | 1.0-4 | 1.0-4 | 1.1-4 | 68.5 | .23   |  |

| Descriptor   | Meaning   |
|--------------|---|
| MID          | Material identification number. (Integer > 0)   |
| EX, ETH, EZ  | Young's moduli in the $x$ , $\theta$ , and $z$ directions, respectively. (Real > 0.0)   |
| NUXTH, NUTHZ | Poisson's ratios (coupled strain ratios in the $x\theta$ , $\theta z$ , and $zx$ directions, respectively). (Real)                                  |
| NUZX         | Mass density. (Real)  |
| RHO          | Shear modulus. (Real > 0.0)   |
| AX, ATH, AZ  | Thermal expansion coefficients. (Real)  |
| TREF         | Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark 10. (Real or blank) |
| GE           | Structural element damping coefficient. See Remarks 9. and 11. (Real)   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE (INIT) Case Control command.
3. The few numbers EX, ETH, EZ, and GZX must be present.
4. A warning message will be issued if any value of NUXTH or NUTHZ has an absolute value greater than 1.0.

5. MAT3 materials may only be referenced by the CTRIA6 entry or the PSHLN2 or PLCOMP entries.
6. The mass density RHO will be used to automatically compute mass for the CTRIA6 element.
7. The x-axis lies along the material axis (see [Figure 9-80](#) in the CTRIA6 entry). The  $\theta$ -axis lies in the azimuthal direction. The z-axis is normal to both.
8. The strain-stress relationship is

$$\begin{bmatrix} \epsilon_x \\ \epsilon_\theta \\ \epsilon_z \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} 1 & \frac{\text{NUTHX}}{\text{ETH}} & \frac{\text{NUZX}}{\text{EZ}} & 0 \\ \frac{\text{NUXTH}}{\text{EX}} & 1 & \frac{\text{NUZTH}}{\text{EZ}} & 0 \\ \frac{\text{NUZX}}{\text{EX}} & \frac{\text{NUTHZ}}{\text{ETH}} & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \sigma_{zx} \end{bmatrix} + (T - \text{TREF}) \begin{bmatrix} \text{AX} \\ \text{ATH} \\ \text{AZ} \\ 0 \end{bmatrix}$$

Note that:

$$\frac{\text{NUXTH}}{\text{EX}} = \frac{\text{NUTHX}}{\text{ETH}}$$

$$\frac{\text{NUZX}}{\text{EZ}} = \frac{\text{NUXZ}}{\text{EX}}$$

$$\frac{\text{NUTHZ}}{\text{ETH}} = \frac{\text{NUZTH}}{\text{EZ}}$$

The compliance matrix as shown must be positive definite. This requires that EX, ETH, EZ, and GZX be positive. There are additional restrictions as well, see a standard text on elasticity.

9. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
10. TREF is used for two different purposes:
  - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10. under the MAT1 description.
  - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
11. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).

**MAT4****Heat Transfer Material Properties, Isotropic**

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

**Format:**

| 1    | 2   | 3      | 4    | 5      | 6 | 7     | 8    | 9       | 10 |
|------|-----|--------|------|--------|---|-------|------|---------|----|
| MAT4 | MID | K      | CP   | $\rho$ | H | $\mu$ | HGEN | REFENTH |    |
|      | TCH | TDELTA | QLAT |        |   |       |      |         |    |

**Example:**

|      |    |        |        |       |  |        |  |  |  |
|------|----|--------|--------|-------|--|--------|--|--|--|
| MAT4 | 17 | 6.66-4 | 5.01-5 | 456.2 |  | 1.03-6 |  |  |  |
|------|----|--------|--------|-------|--|--------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification number. (Integer $> 0$ )   |
| K          | Thermal conductivity. (Blank or Real $> 0.0$ )   |
| CP         | Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real $\geq 0.0$ )      |
| $\rho$     | Density. (Real $\geq 0.0$ ; Default = 1.0)   |
| H          | Free convection heat transfer coefficient. (Real or blank)   |
| $\mu$      | Dynamic viscosity. See Remark 2. (Real $> 0.0$ or blank)   |
| HGEN       | Heat generation capability used with QVOL entries. (Real $\geq 0.0$ ; Default = 1.0)               |
| REFENTH    | Reference enthalpy. (Real or blank)  |
| TCH        | Lower temperature limit at which phase change region is to occur. See Remark 6. (Real or blank)    |
| TDELTA     | Total temperature change range within which a phase change is to occur. (Real $\geq 0.0$ or blank) |
| QLAT       | Latent heat of fusion per unit mass associated with the phase change. (Real $> 0.0$ or blank)      |

**Remarks:**

1. The MID must be unique with respect to all other MAT4 and MAT5 entries. MAT4 may specify material properties for any conduction elements as well as properties for a forced convection fluid (see CONVM). MAT4 also provides the heat transfer coefficient for free convection (see CONV).
2. For a forced convection fluid,  $\mu$  must be specified.
3. REFENTH is the enthalpy corresponding to zero temperature if the heat capacity CP is a constant. If CP is obtained through a TABLEM lookup, REFENTH is the enthalpy at the first temperature in the table.

4. Properties specified on the MAT4 entry may be defined as temperature dependent by use of the entry.
5. For RC network solver in thermal analysis, the REFENTH, TCH, TDELTA and QLAT are ignored.
6. The advanced nonlinear elements should be used for phase change to achieve accurate results.

**MAT5**

## Thermal Material Property Definition

Defines the thermal material properties for anisotropic materials.

Format:

| 1    | 2   | 3    | 4   | 5   | 6   | 7   | 8   | 9  | 10 |
|------|-----|------|-----|-----|-----|-----|-----|----|----|
| MAT5 | MID | KXX  | KXY | KXZ | KYY | KYZ | KZZ | CP |    |
|      | RHO | HGEN |     |     |     |     |     |    |    |

Example:

|      |      |      |  |  |      |  |      |     |  |
|------|------|------|--|--|------|--|------|-----|--|
| MAT5 | 24   | .092 |  |  | .083 |  | 0.20 | 0.2 |  |
|      | 2.00 |      |  |  |      |  |      |     |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification number. (Integer > 0)  |
| Kij        | Thermal conductivity. (Real)   |
| CP         | Heat capacity per unit mass. (Real $\geq 0.0$ or blank)                              |
| RHO        | Density. (Real $> 0.0$ ; Default = 1.0)  |
| HGEN       | Heat generation capability used with QVOL entries. (Real $\geq 0.0$ ; Default = 1.0) |

Remarks:

1. The thermal conductivity matrix has the following form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

2. The material identification number may be the same as a MAT1, MAT2, or MAT3 entry but must be unique with respect to other MAT4 or MAT5 entries.
3. MAT5 materials may be made temperature-dependent by use of the MATT5 entry.
4. When used for axisymmetric analysis (CTRIAX6), material properties are represented where:  
 KXX = radial conductivity component  
 KYY = axial conductivity component
5. The KZZ is required if using the advanced nonlinear Shell element with temperature across thickness called out by NLMOPTS,TEMPP,LINE or QUAD option.

**MAT8****Shell Element Orthotropic Material Property Definition**

Defines the material property for an orthotropic material for isoparametric shell elements.

**Format:**

| 1    | 2       | 3     | 4      | 5    | 6    | 7    | 8    | 9     | 10 |
|------|---------|-------|--------|------|------|------|------|-------|----|
| MAT8 | MID     | E1    | E2     | NU12 | G12  | G1Z  | G2Z  | RHO   |    |
|      | A1      | A2    | TREF   | Xt   | Xc   | Yt   | Yc   | S     |    |
|      | GE      | F12   | STRN   |      |      |      |      |       |    |
|      | "HFAIL" | HF1   | HF2    | HF3  | HF4  | HF10 | HF11 |       |    |
|      | "HTAPE" | HT1   | HT2    | HT3  | HT4  | HT5  | HT6  | HT10  |    |
|      |         | H11   | HT12   |      |      |      |      |       |    |
|      | "HFABR" | HFB1  | HFB2   | HFB3 | HFB4 | HFB5 | HFB6 | HFB10 |    |
|      |         | HFB11 | HFBT12 |      |      |      |      |       |    |

**Example:**

|      |         |       |       |       |       |       |       |       |  |
|------|---------|-------|-------|-------|-------|-------|-------|-------|--|
| MAT8 | 171     | 30.+6 | 1.+6  | 0.3   | 2.+6  | 3.+6  | 1.5+6 | 0.056 |  |
|      | 28.-6   | 1.5-6 | 155.0 | 1.+4  | 1.5+4 | 2.+2  | 8.+2  | 1.+3  |  |
|      | 1.-4    |       | 1.0   |       |       |       |       |       |  |
|      | "HFAIL" | 1010. | 1020. | 33.   | 34.   | 1011. | 1012. |       |  |
|      | "HTAPE" | 2001. | 2002. | 35.   | 36.   | 2003. | 1.0   | 1004. |  |
|      |         | 1007. | 1008. |       |       |       |       |       |  |
|      | "HFABR" | 3001. | 3002. | 3003. | 3004. | 1005. | 1005. | 1006. |  |
|      |         | 1007. | 1008. |       |       |       |       |       |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material identification number. Referenced on a PSHELL, PCOMP or PCOMPG entry only. (0 < Integer < 100,000,000)   |
| E1         | Modulus of elasticity in X material coordinate direction, also defined as the fiber direction or 1-direction. (Real > 0.0)  |
| E2         | Modulus of elasticity in Y material coordinate direction, also defined as the matrix direction or 2-direction. (Real > 0.0)   |
| NU12       | Poisson's ratio ( $\nu_{21} = \varepsilon_2 / \varepsilon_1$ for uniaxial loading in 1-direction). Note that $\nu_{21} = \varepsilon_1 / \varepsilon_2$ for uniaxial loading in 2-direction is related to $\nu_{12}$ , $E_1$ , and $E_2$ by the relation $\nu_{12}E_2 = \nu_{21}E_1$ . (Real) |
| G12        | In-plane shear modulus. (Real > 0.0; Default = 0.0)   |
| G1Z        | Transverse shear modulus for shear in 1-Z plane. (Real $\geq 0.0$ or blank; Default implies infinite shear modulus.)  |

| Descriptor | Meaning   |
|------------|---|
| G2Z        | Transverse shear modulus for shear in 2-Z plane. (Real $\geq 0.0$ or blank; Default implies infinite shear modulus.)  |
| RHO        | Mass density. (Real)  |
| Ai         | Thermal expansion coefficient in i-direction. (Real)  |
| TREF       | Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 4. and 5. (Real or blank)   |
| Xt, Xc     | Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. See the FT field on the PCOMP/PCOMPG entry. (Real $> 0.0$ ; Default value for Xc is Xt.) |
| Yt, Yc     | Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if failure index is desired. (Real $> 0.0$ ; Default value for Yc is Yt.)  |
| S          | Allowable stress or strain for in-plane shear. See the FT field on the PCOMP/PCOMPG entry. (Real $> 0.0$ )  |
| GE         | Structural damping coefficient. See Remarks 4. and 6. (Real)  |
| F12        | Interaction term in the tensor polynomial theory of Tsai-Wu. Required if failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See the FT field on the PCOMP/PCOMPG entry. (Real)                     |
| STRN       | For the maximum strain theory only (see STRN in PCOMP/PCOMPG entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables. (Real = 1.0 for strain allowables; blank (Default) for stress allowables.)              |
| “HFFAIL”   | Keyword indicating that Hashin failure criterion, to calculate its four failure indices.  |
| HF1        | Maximum fiber tensile stress, no default  |
| HF2        | Maximum fiber compressive stress, default=HF1   |
| HF3        | Maximum matrix tensile stress, no default   |
| HF4        | Maximum matrix compressive stress, default=HF3  |
| HF10       | Maximum in-plane shear stress, no default   |
| HF11       | Maximum transverse shear stress, default=HF10   |
| “HTAPE”    | Keyword indicating that Hashin-Tape criterion, a variant of the Hashin criterion, adapted for tape type of materials are calculated.  |
| HT1        | Maximum tape fiber tensile stress, no default   |
| HT2        | Maximum tape fiber compressive stress, default=HT1  |
| HT3        | Maximum tape cross-fiber tensile stress, no default   |
| HT4        | Maximum tape cross-fiber compressive stress, default=HT3  |
| HT5        | Maximum fiber tensile stress for matrix compression; Required if HT6=1.0, otherwise not used.   |

| Descriptor | Meaning  |
|------------|--|
| HT6        | Contribution factor for HT5 (Real, 0.0 or 1.0, default = 0.0)  |
| HT10       | Maximum in-plane shear stress, no default  |
| HT11       | Maximum transverse shear stress, default=HT10  |
| HT12       | Maximum z-x transverse shear stress, default=HT11  |
| "HFABR"    | Keyword indicating that Hashin-Fabric criterion, a variant of the Hashin criterion, adapted for fabric type of materials are calculated. |
| HFB1       | Maximum first fiber tensile stress, no default   |
| HFB2       | Maximum first fiber compressive stress, default=HFB1   |
| HFB3       | Maximum second cross-fiber tensile stress, no default  |
| HFB4       | Maximum second cross-fiber compressive stress, default=HFB3  |
| HFB5       | Maximum thickness tensile stress, no default   |
| HFB6       | Maximum thickness compressive stress, default=HFB5   |
| HFB10      | Maximum in-plane shear stress, no default  |
| HFB11      | Maximum transverse shear stress, default=HFB10   |
| HFB12      | Maximum z-x transverse shear stress, default=HFB11   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. If G1Z and G2Z values are specified as zero or blank, then transverse shear flexibility calculations will not be performed, which is equivalent to zero shear flexibility (i.e., infinite shear stiffness). If MAT8 is referenced by Advanced Nonlinear Element, G1Z and G2Z must be positive.
3. An approximate value for G1Z and G2Z is the in-plane shear modulus G12. If test data are not available to accurately determine G1Z and G2Z for the material and transverse shear calculations are deemed essential; the value of G12 may be supplied for G1Z and G2Z. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed in the TEMPERATURE Case Control command.
4. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP/PCOMPG entry. Xc and Yc are also used but not required.
5. TREF and GE are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
6. TREF is used in two different ways:
  - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-105](#) in Remark 10. in the MAT1 description.

- In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.
7. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
  8. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See [Parameters, 783](#).
  9. If MAT8 with Hashin is used in SOL 400 with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

**MAT8A****Orthotropic Failure Material Properties - SOL700**

Defines the failure properties for an orthotropic material for shell elements. Used in SOL700 only.

**Format:**

| 1     | 2     | 3     | 4     | 5     | 6     | 7       | 8    | 9 | 10 |
|-------|-------|-------|-------|-------|-------|---------|------|---|----|
| MAT8A | MID   | FT    | NV    | S     | ALPHA | TRSFAIL | F12  |   |    |
|       | XT    | XC    | YT    | YC    | PFD   | VALUE   | PFDS |   |    |
|       | FBTEN | FBCOM | MXTEN | MXCOM | MXSHR |         |      |   |    |
|       | PRDFT | PRDFC | PRDMT | PRDMC | PRDSH |         |      |   |    |

**Example:**

|       |       |          |        |         |       |       |  |  |  |
|-------|-------|----------|--------|---------|-------|-------|--|--|--|
| MAT8A | 7     | COMBINAT |        | 100.    |       |       |  |  |  |
| +     | 200.0 | 150.0    | 100.0  | 110.0   | STEPS | 200.0 |  |  |  |
| +     | CHANG | STRSS    | MODSAI | MODTSAI | STRSS |       |  |  |  |
| +     |       |          |        |         |       |       |  |  |  |
| +     |       |          |        |         | 0011  |       |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material identification. See Remark 1. (Integer > 0, Required.)   |
| FT         | Failure theory to be used to test whether the element layer fails. (Character; default=blank)                                       |
|            | Blank No failure  |
|            | HILL Tsai-Hill theory   |
|            | TSAI Tsai-Wu theory   |
|            | MODTSAI Modified Tsai-Wu theory   |
|            | STRSS Maximum stress  |
|            | CHANG Chang-Chang theory  |
|            | COMBINAT Combination (See Remark 8.)  |
|            | HASHIN Hashin theory (See Remark 2.)  |
| NV         | Number of additional history variables for a user model. See Remark 8. Currently it is not available. (0 < Integer < 10; default=0) |
| S          | Failure stress for in-plane shear. See Remark 4. (Real > 0.0; required.)  |
| ALPHA      | Nonlinear shear coefficient. See Remark 5. (Real $\geq$ 0.0; default=0.0)   |
| TRSFAIL    | Transverse shear failure. (Character; default=SUBL)   |

| Descriptor | Meaning |  |
|------------|---------|--|
|            | ELEM    | Failure if element fails   |
|            | SUBL    | Failure if sublayer fails  |
| F12        |         | Interaction term in Tsai-Wu theory. (Real, default=0.0)  |
| XT         |         | Tensile failure stress in the large structural direction. See Remark 4. (Real $\geq 0.0$ ; default=0.0)                      |
| XC         |         | Compressive failure stress in the large structural direction. See Remark 4. (Real $\geq 0.0$ ; default=0.0)                  |
| YT         |         | Tensile failure stress in the lateral direction. See Remark 4. (Real $\geq 0.0$ ; default=0.0)                               |
| YC         |         | Compressive failure stress in the lateral direction. See Remark 4. (Real $\geq 0.0$ ; default=0.0)                           |
| PFD        |         | Post-failure degradation model. See Remark 9. (Character; default=STEPS)   |
|            | STEPS   | Degrade stresses by time steps   |
|            | TIME    | Degrade stresses by time   |
|            | VELOC   | Degrade stresses by velocity   |
| VALUE      |         | Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Real $> 0$ ; default=100.0) |
| PFDST      |         | Post-failure degradation start. See Remark 9. (Character; default=INDV)  |
|            | INDV    | Stresses are degraded per distinct failure mode.   |
|            | ALL     | Stresses are degraded if all elastic constants are zero.   |
| FBTEN      |         | Tensile failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)             |
| FBCON      |         | Compressive failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)         |
| MXTEN      |         | Tensile failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)            |
| MXCOM      |         | Compressive failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)        |
| MXSHR      |         | Shear failure modes for individual failure definition. See Remark 6. (Character; required.)                                  |
| PRDFT      |         | Property degradation due to fiber-tension failure. See Remark 7. (Integer; default=1111)                                     |
| PRDFC      |         | Property degradation due to fiber-compression failure. See Remark 7. (Integer; default=1010)                                 |
| PRDMT      |         | Property degradation due to matrix-tension failure. See Remark 7. (Integer; default=0110)                                    |

| Descriptor | Meaning   |
|------------|---|
| PRDMC      | Property degradation due to matrix-compression failure. See Remark 7. (Integer; default=0110) |
| PRDSH      | Property degradation due to in-plane shear failure. See Remark 7. (Integer; default=0001)     |

**Remarks:**

1. The material number must refer to a MAT8 material definition.
2. If a failure theory is selected other than COMBINAT, the theory defines the following failure modes:

|         |   |
|---------|---|
| CHANG   | Fiber tension, matrix tension/compression               |
| HILL    | All modes   |
| TSAI    | All modes   |
| MODTSAI | Matrix tension/compression                              |
| STRSS   | All modes   |
| HASHIN  | Fiber tension/compression<br>Matrix tension/compression |

For an element to fail completely, both fiber and matrix in all sublayers must fail.

3. This material model can only be referenced from a PCOMP entry.
4. Failure stresses are required if a failure theory is selected.
5. ALPHA is used for all failure theories to define a nonlinear stress-strain relation.
6. The individual failure modes are defined according to the corresponding mode in the theory as listed under FT. To be relevant, the theory must define the failure mode (see Remark 2.). You must enter data if FT is set to COMBINAT.
7. The property degradation rules due to the various failure modes are listed below:

| Material Constant | Failure Mode |            |             |             |       |
|-------------------|--------------|------------|-------------|-------------|-------|
|                   | Fiber Tens   | Fiber Comp | Matrix Tens | Matrix Comp | Shear |
| E1                | X            | X          |             |             |       |
| E2                | X            |            | X           | X           |       |
| V12               | X            | X          | X           | X           |       |
| G12               | X            |            |             |             | X     |

The Poisson's ratio Nu21 is treated the same as Nu12.

To override the default model, an integer value is defined as a packed word in the following order:

- 1 Denotes property degradation.
- 0 Denotes no degradation.

The last five fields of the MAT8A Bulk Data entry are input for the user to specify the degradation behavior for each mode of failure.

8. NV is required input and NV new user variables are automatically created. User variables for sublayers are used on restart and archive output. Refer to them as USRnLxx where n is the user ID and xx is the sublayer number. The user variables are available in the EXCOMP user subroutine. The values S, XT, XC, YT, and YC are also required input when FT is set to USER. For the model USER1, the routine EXCOMP1 has to be used. Currently, user routine is not available.
9. The PFD entry indicates how the stresses are degraded to zero. The PFDST indicates when the stresses start to degrade.  
Using ALL means that degradation starts when all material constants (E1, E2, V12, G12) are degraded to zero as specified by the FT entry and the property degradation rules. Note that property degradation means that the stress increments are zero but that the stresses degrade according to PFD.  
INDV means that stress degradation starts for the fiber stress if E1=0.0, for matrix stress if E2=0.0, and for shear stress if G12=0.0.
10. Any failure theory introduces five additional sublayer variables. The PFDST entry introduces three additional variables. The number of user variables is defined by NV.

**MAT8F****Shell Element Orthotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT8 entry fields via TABLEDi entries.

**Format:**

| 1     | 2   | 3  | 4  | 5    | 6   | 7   | 8   | 9 | 10 |
|-------|-----|----|----|------|-----|-----|-----|---|----|
| MAT8F | MID | E1 | E2 | NU12 | G12 | G1Z | G2Z |   |    |
|       |     |    |    |      |     |     |     |   |    |
|       | GE  |    |    |      |     |     |     |   |    |

**Example:**

|       |    |  |  |  |  |  |  |  |  |
|-------|----|--|--|--|--|--|--|--|--|
| MAR8F | 76 |  |  |  |  |  |  |  |  |
|       |    |  |  |  |  |  |  |  |  |
|       | 97 |  |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on MAT8 entry. (Integer > 0)  |
| E1         | Identification number of a TABLEDi entry for the Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Integer > 0 or blank) |
| E2         | Identification number of a TABLEDi entry for the Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Integer > 0 or blank)     |
| NU12       | Identification number of a TABLEDi entry for Poisson's ratio. (Integer > 0 or blank)   |
| G12        | Identification number of a TABLEDi entry for the in-plane shear modulus. (Integer > 0 or blank)  |
| G1Z        | Identification number of a TABLEDi entry for the transverse shear modulus for shear in 1-Z plane. (Integer > 0 or blank)   |
| G2Z        | Identification number of a TABLEDi entry for the transverse shear modulus for shear in 2-Z plane. (Integer > 0 or blank)   |
| GE         | Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)  |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT8 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT8 then GE table must be blank or 0.

## MAT9 Solid Element (and Shell Element for SOL 600 only) Anisotropic Material Property Definition

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

Format:

| 1    | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|------|------|------|------|------|------|------|------|------|----|
| MAT9 | MID  | G11  | G12  | G13  | G14  | G15  | G16  | G22  |    |
|      | G23  | G24  | G25  | G26  | G33  | G34  | G35  | G36  |    |
|      | G44  | G45  | G46  | G55  | G56  | G66  | RHO  | A1   |    |
|      | A2   | A3   | A4   | A5   | A6   | TREF | GE   |      |    |
|      | GE11 | GE12 | GE13 | GE14 | GE15 | GE16 | GE22 | GE23 |    |
|      | GE24 | GE25 | GE26 | GE33 | GE34 | GE35 | GE36 | GE44 |    |
|      | GE45 | GE46 | GE55 | GE56 | GE66 |      |      |      |    |

Example:

|      |       |       |  |       |       |       |      |       |  |
|------|-------|-------|--|-------|-------|-------|------|-------|--|
| MAT9 | 17    | 6.2+3 |  |       |       |       |      | 6.2+3 |  |
|      |       |       |  |       | 6.2+3 |       |      |       |  |
|      | 5.1+3 |       |  | 5.1+3 |       | 5.1+3 | 3.2  | 6.5-6 |  |
|      | 6.5-6 |       |  |       |       | 125.  | .003 |       |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification number. (Integer > 0)  |
| Gij        | Elements of the 6 x 6 symmetric material property matrix in the material coordinate system. (Real)   |
| RHO        | Mass density. (Real)   |
| Ai         | Thermal expansion coefficient. (Real)  |
| TREF       | Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark 7. (Real or blank) |
| GE         | Structural element damping coefficient. See Remarks 6. and 8. (Real)   |
| GEij       | Structural damping matrix. See Remark 9. (Real)  |

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

2. MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In nonlinear static analysis (e.g., SOL 106), linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density RHO will be used to compute mass in a structural dynamics problem automatically.
4. The third continuation entry is optional.
5. The subscripts 1 through 6 refer to x, y, z, xy, yz, and zx of the material coordinate system (see the CORDM field on the PSOLID entry description). The stress-strain relationship is

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{Bmatrix} G_{11} & G_{12} & G_{13} & G_{14} & G_{15} & G_{16} \\ & G_{22} & G_{23} & G_{24} & G_{25} & G_{26} \\ & & G_{33} & G_{34} & G_{35} & G_{36} \\ & & & G_{44} & G_{45} & G_{46} \\ & & & & G_{55} & G_{56} \\ & & & & & G_{66} \end{Bmatrix}_{\text{symmetric}} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix} (T - TREF)$$

6. The damping coefficient GE is given by

$$GE = \frac{2.0 \cdot C}{C_0}$$

7. TREF is used in two different ways:

- In nonlinear static analysis (e.g., SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-105](#) in Remark 10. in the MAT1 description.
- In all solutions except nonlinear static analysis, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.

8. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
9. Defines structural damping matrix data for advanced composites. If the GEij values are present, the Nastran will ignore the GE value given on the third continuation entry field (8) and use the given GEij values. The rational of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMPLS MIDi entries must refer to MAT9 entries.

The damping is applied in element material routines by forming the product GEij \* Gij when forming the K4 damping matrix in EMG.

## MAT9F Solid Element Anisotropic Frequency Dependence

Specifies frequency-dependent material properties on MAT9 entry fields via TABLEDi entries.

Format:

| 1     | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|------|------|------|------|------|------|------|----|
| MAT9F | MID  | G11  | G12  | G13  | G14  | G15  | G16  | G22  |    |
|       | G23  | G24  | G25  | G26  | G33  | G34  | G35  | G36  |    |
|       | G44  | G45  | G46  | G55  | G56  | G66  |      |      |    |
|       |      |      |      |      |      |      | GE   |      |    |
|       | GE11 | GE12 | GE13 | GE14 | GE15 | GE16 | GE22 | GE23 |    |
|       | GE24 | GE25 | GE26 | GE33 | GE34 | GE35 | GE36 | GE44 |    |
|       | GE45 | GE46 | GE55 | GE56 | GE66 |      |      |      |    |

Example:

|       |     |   |  |   |   |   |  |   |  |
|-------|-----|---|--|---|---|---|--|---|--|
| MAT9F | 101 | 5 |  |   |   |   |  | 5 |  |
|       |     |   |  |   | 5 |   |  |   |  |
|       |     |   |  | 5 |   | 5 |  |   |  |
|       |     |   |  |   |   |   |  |   |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)      |
| Gij        | Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)                |
| GE         | Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)  |
| GEij       | Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank) |

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value in a particular field of the MAT9 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT9 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT9 then GE table must be blank or 0. For any GEij=0.0 on corresponding MAT9, the corresponding GEij table must be blank or 0.

**MAT10****Fluid Material Property Definition**

Defines material properties for fluid elements in coupled fluid-structural analysis.

**Format:**

| 1     | 2   | 3    | 4   | 5 | 6  | 7     | 8 | 9 | 10 |
|-------|-----|------|-----|---|----|-------|---|---|----|
| MAT10 | MID | BULK | RHO | C | GE | ALPHA |   |   |    |

**Example:**

|       |     |       |       |  |  |  |  |  |  |
|-------|-----|-------|-------|--|--|--|--|--|--|
| MAT10 | 103 | 0.656 | 0.011 |  |  |  |  |  |  |
|-------|-----|-------|-------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Material identification number. (Integer > 0)  |
| BULK       | Bulk modulus. (Real > 0.0)   |
| RHO        | Mass density. (Real > 0.0)   |
| C          | Speed of sound. (Real > 0.0)   |
| GE         | Fluid element damping coefficient. (Real)  |
| ALPHA      | Normalized admittance coefficient for porous material. See Remark 7. (Real or blank) |

**Remarks:**

1. MAT10 is referenced, with MID, by the PSOLID entry only.
2. The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT9, and MAT10 entries.
3. The mass density RHO will be used to compute the mass automatically.
4. BULK, RHO, and C are related by

$$\text{BULK} = C^2 \cdot \text{RHO}$$

Two out of the three must be specified, and the other will be calculated according to this equation. If all three are specified and are inconsistent in values, the supplied values of BULK and RHO are used in the computations.

5. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$ , by 2.0.
6. If PARAM,W4FL is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
7. If a value of ALPHA is entered, BULK RHO and GE may have negative values.
8. The value defined in the ALPHA field always defines the normalized admittance coefficient for porous material but it is differently interpreted depending on the value defined in the FCNT field of the referencing PSOLID entry.

- a. If the MAT10 entry is referenced in a PSOLID entry where FFLUID option is selected, the value defined for ALPHA is considered as the normalized admittance coefficient calculated at unit circular excitation frequency ( $\omega = 1$ ). Its value will be automatically calculated by the program, at each excitation frequency, considering the current circular excitation frequency as scaling factor.
  - b. If the MAT10 entry is referenced in a PSOLID entry where PFLUID option is selected, the value defined for ALPHA has no special meaning but it is only the normalized admittance coefficient calculated by the user at the most appropriate excitation frequency (defined in order to have good results in the frequency range of interest).
9. For Poro-Elastic Material, PEM, user is expected to provide adiabatic BULK modulus for fluid-phase of PEM which is equal to GAMMA, fluid ratio of specific heats (see MATPE1 for detailed), times isothermal BULK modulus. Isothermal BULK modulus is BULK modulus at constant temperature for fluid itself.
  10. When used in conjunction with PSLDN1, only pyramid elements can be associated with the MAT10 fluid property. CHEXA, CPENTA, CTETRA elements using MAT10 are not supported with PSLDN1.
  11. The terms of the fluid mass matrix are not multiplied by the value of PARAM and WTMASS.

**MAT10F****Fluid Frequency Dependence**

Specifies frequency-dependent material properties on MAT10 entry fields via TABLED<sub>i</sub> entries.

**Format:**

| 1      | 2   | 3    | 4   | 5 | 6  | 7     | 8 | 9 | 10 |
|--------|-----|------|-----|---|----|-------|---|---|----|
| MAT10F | MID | BULK | RHO | C | GE | ALPHA |   |   |    |

**Example:**

|       |     |     |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|
| MAT10 | 103 | 666 |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Material property identification number that matches the identification number on MAT10 entry. (Integer > 0)                      |
| BULK       | Identification number of a TABLED <sub>i</sub> entry for the Bulk modulus. (Integer > 0 or blank)                                 |
| RHO        | Identification number of a TABLED <sub>i</sub> entry for the mass density. Used only in SOL108. (Integer > 0 or blank)            |
| C          | Identification number of a TABLED <sub>i</sub> entry for the speed of sound. Used only in SOL108. (Integer > 0 or blank)          |
| GE         | Identification number of a TABLED <sub>i</sub> entry for the element structural damping coefficient. (Integer > 0 or blank)       |
| ALPHA      | Identification number of a TABLED <sub>i</sub> entry for the normalized admittance coefficient for porous material. See Remark 4. |

**Remarks:**

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT10 entry referenced in field 2. The value in a particular field of the MAT10 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT10 entry.
2. Table references must be present for each item that is frequency dependent.
3. When ALPHA on MAT10 is specified it is multiplied by the excitation frequency in radians/unit time. If F(ALPHA) is specified then the value supplied by F(ALPHA) will take precedence and ALPHA WILL NOT BE MULTIPLIED by the excitation frequency.

**MATDEUL**

## General Constitutive Model to be Used for the Eulerian Materials - SOL 700

Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame. Used in SOL 700 only.

Format:

| 1       | 2     | 3     | 4   | 5   | 6       | 7   | 8   | 9 | 10 |
|---------|-------|-------|-----|-----|---------|-----|-----|---|----|
| MATDEUL | MID   | RHO   | EID | SID | YID     | FID | PID |   |    |
|         | BULKL | BULKQ |     |     | BULKTYP |     |     |   |    |

Example:

|         |    |       |     |     |     |     |     |  |  |
|---------|----|-------|-----|-----|-----|-----|-----|--|--|
| MATDEUL | 22 | 3000. | 100 | 109 | 307 | 308 | 402 |  |  |
|---------|----|-------|-----|-----|-----|-----|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Unique material number. (Integer > 0; Required)   |
| RHO        | Density. (Real > 0.0; Required)   |
| EID        | Number of an EOSxxx entry defining the pressure/density characteristic of the material. (Integer > 0; Required)       |
|            | EOSGAM      Gamma Law Gas Equation of State   |
|            | EOSIG      Ignition and Growth Equation of State  |
|            | EOSJWL      JWL Explosive Equation of State   |
|            | EOSMG      Mie-Gruneisen Equation of State  |
|            | EOSPOL      Polynomial Equation of State  |
|            | EOSTAIT      Tait Equation of State   |
| SID        | Number of a SHRxxy entry defining the shear properties of the material. (Integer $\geq 0$ ; Hydrodynamic shear model) |
|            | SHREL      Elastic Shear Model  |
|            | SHRPOL      Polynomial Shear Mode   |
| YID        | Number of a YLDxxx entry defining the yield model for the material. (Integer $\geq 0$ ; Hydrodynamic yield model)     |
|            | YLDHY      Hydrodynamic Yield Model   |
|            | YLDJC      Johnson-Cook Yield Model   |
|            | YLDMC      Mohr-Coulomb Yield Model   |
|            | YLDMSS      Multi-surface yield model for Snow  |
|            | YLDPOL      Polynomial Yield Model  |
|            | YLDRPL      Rate Power Law Yield Model  |

| Descriptor | Meaning   |
|------------|---|
| YLDHG      | Steinberg-Guinan Yield  |
| YLDHM      | Tanimura-Mimura Yield Model   |
| YLDVM      | von Mises Yield Model   |
| YLDZA      | Zerilli-Armstrong Yield Model   |
| FID        | Number of a FAILMPS entry defining the failure model for the material. (Integer $\geq 0$ ; no failure)          |
|            | FAILMPS Maximum Plastic Strain Failure Model  |
| PID        | Number of a PMINC entry defining the spallation characteristics of the material. See Remark 3. (Integer $> 0$ ) |
| BULKL      | Linear bulk-viscosity coefficient. (Real $\geq 0.0$ ; 0.0)  |
| BULKQ      | Quadratic bulk-viscosity coefficient. (Real $> 0.0$ ; 1.0)  |
| BULKTYP    | Bulk viscosity type. (Character, DYNA)  |
|            | DYNA Standard DYNA3D model.   |
|            | DYTRAN Enhanced DYNA model.   |

#### Remarks:

1. If YID is blank or zero, a hydrodynamic yield model is used.
2. If the TYPE field on the PEULER entry is set to HYDRO, then YID is either blank or references a YLDHY entry, and SID is blank. Conversely, if the TYPE field is set to STRENGTH, a nonhydrodynamic yield model is specified.
3. If no PMINC entry is referenced, a minimum pressure of zero is assumed for the Eulerian elements. The PMINC entry will be ignored when a cavitation model through the EOSTAIT entry has been given.

**MATDIGI****Material Digimat**

Defines material data for the advanced composites with Digimat from e-Xstream engineering (SOL 400 only). For more information about e-Xstream engineering and Digimat, please contact support@e-xstream.com or consult <http://www.e-xstream.com/>

**Format:**

| 1       | 2   | 3    | 4 | 5 | 6 | 7 | 8   | 9 | 10 |
|---------|-----|------|---|---|---|---|-----|---|----|
| MATDIGI | MID | UDID |   |   |   |   | RHO |   |    |

**Example:**

|         |   |    |  |  |  |  |       |  |  |
|---------|---|----|--|--|--|--|-------|--|--|
| MATDIGI | 5 | 10 |  |  |  |  | 3000. |  |  |
|---------|---|----|--|--|--|--|-------|--|--|

| Descriptor | Meaning                                       |
|------------|---|
| MID        | Material identification number. (Integer > 0) |
| UDID       | References UDNAME entry (Required, Integer>0) |
| RHO        | Density (Real>0.0 or blank)                   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. SOL400 uses only the MID, UDID and RHO entries. All other entries are ignored by SOL400.
3. The entry UDSESV is required input to define the number of state variables which will be used by MATDIGI. The number of state variables (NSTATS) must be larger than 1.
4. License feature NASTRAN\_DIGIMAT must be present to use this feature. For parallel computations, NASTRAN\_DIGIMAT\_DMP must be present.
5. The use of MATDIGI is limited to shell and solid elements.
6. MATDIGI should not be used in SOL 400 perturbation steps.

**MATEP****Elasto-Plastic Material - SOL 400**

Elasto-plastic material properties for SOL 400 only.

**Format:** Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

| 1     | 2          | 3                   | 4        | 5         | 6         | 7         | 8         | 9     | 10 |
|-------|------------|---------------------|----------|-----------|-----------|-----------|-----------|-------|----|
| MATEP | MID        | FORM                | Y0       | FID       | RYIELD    | WKHARD    |           | H     |    |
|       | “Reffect”  | OPTION              | RTID     | C         | P         |           |           |       |    |
|       | “Aniso”    | N/A                 | R11 or M | R22 or C1 | R33 or C2 | R12 or C3 | R23 or C6 | R31   |    |
|       | “Press”    | OPTION              | ALPHS    | BETA      |           |           |           |       |    |
|       | “Chaboche” | R0                  | Rinf     | B         | C         | Gam       | Kap       | N     |    |
|       |            | Qm                  | $\mu$    | $\eta$    |           |           |           |       |    |
|       | “JhCook”   | A                   | B        | N         | C         | M         | Tmelt     | Troom |    |
|       |            | $\varepsilon_0$ DOT |          |           |           |           |           |       |    |
|       | “YldOpt”   | OPTION              |          |           |           |           |           |       |    |
|       | “VParam”   | Nvp                 | Vp1      | Vp2       |           |           |           |       |    |
|       | “IMPCREEP” | VMISES              |          |           |           |           |           |       |    |

**Example:**

|       |       |       |     |     |  |      |  |  |  |
|-------|-------|-------|-----|-----|--|------|--|--|--|
| MATEP | 100   | TABLE |     | 20  |  | HILL |  |  |  |
|       | ANISO |       | 1.1 | 0.9 |  | 1.02 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MAT1, MAT2, MATORT, MAT8 or MAT9 entry.<br>(Integer > 0).  |
| FORM       | Selects a form of stress-plastic strain function to be specified (Character):<br><br>SLOPE                          Defines nonzero H. (Default)<br><br>TABLE                          Defines the function using TABLES1, TABLEST or TABL3D entries. Units of table are unspecified<br><br>PERFECT                        Defines perfectly plastic material (H is zero or blank and FID blank).<br><br>See Remarks 2. |
| Y0         | Initial yield stress $Y_0$ or hydrostatic stress for Mohr-Coulomb materials. See Remark 2.<br>(Real > 0 or blank)   |
| FID        | Identification number of TABLES1 or TABL3D entry. See Remarks 2., 3., and 4. (Integer > 0 or blank)   |

| Descriptor | Meaning   |
|------------|---|
| RYIELD     | Enter one of the following yield criteria rules. (Character; Default = VMISES) See Remarks 12.<br>VMISES von Mises yield criteria (Default)<br>HILL Hill's 1948 yield criteria<br>BARLAT Barlat's 1991 yield criteria<br>LINMOHR Linear Mohr-Coulomb yield criteria<br>PBLMOHR Parabolic Mohr-Coulomb yield criteria<br>IMPCREEP Implicit creep model combining both plasticity and creep, von Mises yield criteria.  |
| WKHARD     | Selects a hardening rule defined by various work-hardening rules (Character). See Remarks 12.<br>ISOTROP Isotropic hardening. (Default)<br>KINEM Kinematic hardening.<br>COMBINE Combination between kinematic and isotropic hardening. (Default kinematic fraction=0.5).<br>CHABOCHE Caboche formulation - like combined (see MSC Nonlinear User Guide (SOL 400)) for details of this model).<br>JHCOOK For Johnson-Cook formulation (see MSC Nonlinear User Guide (SOL 400)) for details of this model).<br><br>See Remark 5. |
| H          | Workhardening slope. See Remarks 2., 3., and 4. (Real $\geq 0$ ; Default = 0)   |
| "Reffect"  | A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABL3D is entered and one of the independent variables is strain rate, this field and its options should not be specified.   |
| OPTION     | Selects an option for strain-rate dependent yield stress (Character):<br>TABLE Specifies that TABLES1 input used to define rate-dependency. (Default)<br>COWPER Specifies the Cowper and Symonds rate-dependency model. See Remark 6.   |
| RTID       | TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.   |
| C          | Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)  |
| P          | Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)  |

| Descriptor | Meaning  |
|------------|--|
| "Aniso"    | A keyword signifying that the following data ( $R_{ij}$ ) pertain to the Hill's ( $R_{ij}$ ) or Barlat's ( $M$ , $C_i$ ) yield criteria. an isotropic material option. See Remark 7. and MSC Nonlinear User Guide (SOL 400) (specify only if RYIELD = HILL or BARLAT). |
| $R_{ij}$   | Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real > 0; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill) or   |
| M          | Barlat M coefficient   |
| C1         | Barlat C1 coefficient  |
| C2         | Barlat C2 coefficient  |
| C3         | Barlat C3 coefficient  |
| C6         | Barlat C6 coefficient  |
| "Press"    | A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield = LinMohr or PblMohr cracking is to be simulated. See Remarks 8. and 9.  |
| OPTION     | Selects an option for pressure-dependent yield criteria. (Character):  |
| LIN        | Linear Mohr-Coulomb model. (Default)   |
| PBL        | Parabolic Mohr-Coulomb model.  |
| ALPHA      | Specifies the parameter alpha for linear Mohr-Coulomb model. (Real > 0 or blank)   |
| BETA       | Specifies the parameter beta for parabolic Mohr-Coulomb model. (Real > 0 or blank)   |
| "Chaboche" | A keyword specifying the following data pertains to the Chaboche model. (Enter only if WKHARD=CHABOCHE.)   |
| R0         | R0 for isotropic hardening (Real)  |
| Rinf       | Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization) (Real)   |
| B          | Coefficient B for isotropic hardening (Real)   |
| C          | C coefficient C for kinematic hardening (Real)   |
| Gam        | Gamma coefficient for kinematic hardening (Real)   |
| Kap        | Kappa value for viscosity model (Real)   |
| N          | Coefficient N for viscosity model (Real)   |
| Qm         | Coefficient Qm for isotropic hardening (Real)  |
| $\mu$      | Coefficient $\mu$ for isotropic hardening (Real)   |
| $\eta$     | Coefficient $\eta$ to introduce progressive memory (Real)  |
| "JhCook"   | A keyword specifying the following data pertains to the Johnson-Cook model (see the MSC Nonlinear User Guide (SOL 400) for more details. Enter only if WKHARD = JHCOOK).   |

| Descriptor                | Meaning   |
|---------------------------|---|
| A                         | Coefficient A. (Real)   |
| B                         | Coefficient B. (Real)   |
| N                         | Exponent n. (Real)  |
| C                         | Coefficient C. (Real)   |
| M                         | Exponent m. (Real)  |
| Tmelt                     | Melting temperature. (Real)   |
| Troom                     | Ambient temperature. (Real)   |
| $\dot{\varepsilon}_0$ Dot | Reference strain rate. (Real)   |
| "YldOpt"                  | A keyword specifying one of the following yield models will be used.<br>Select a yield option from the list below:  |
| OPTION                    | Specifies a yield option. (Character):  |
|                           | IMCREEP      Implicit creep model, both plasticity and creep with von Mises yield criteria. -user subroutine UVSCPL. MATUDS is required.                              |
| "VParam"                  | A keyword specifying that the number of viscoplastic parameters is to follow. Up to two viscoelastic parameters are to be entered (used only when RYIELD = IMPCREEP). |
| Nvp                       | Number of viscoplastic parameters. (Integer must be 0, 1, or 2)   |
| Vp1, Vp2                  | A list of viscoplastic parameters. (Real, Default = 0.0)  |
| "IMPCREEP"                | A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used only when RYIELD = IMPCREEP).                       |
| VMISES                    | Equivalent (von Mises) tensile yield stress. Overrides any similar entry made for this material elsewhere. (Real, no Default)   |

### Remarks:

- Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
- If FORM = SLOPE, the one defines the initial yield stress with  $Y_0$  and a single workhardening slope is entered as  $H \cdot H = \frac{\partial \bar{\sigma}}{\partial \dot{\varepsilon}^p}$ . In a uniaxial test, , the  $\dot{\varepsilon}_0$  is the strain at which yielding first occurs is often considered the 0.2% offset strength.

If FORM = TABLE, then the table ID is entered using FID field,  $Y_0$  is not used. The table can be entered as TABLES1 or TABL3D.

When TYPE = 1 on the TABLES1 option, one is defining  $\bar{\sigma} = \bar{\sigma}(\dot{\varepsilon})$  as shown in [Figure 9-107](#).

When TYPE = 2 on the TABLES1 or TABL3D option, one is defining  $\bar{\sigma} = \bar{\sigma}(\dot{\varepsilon}^p)$  as shown in [Figure 9-108](#).

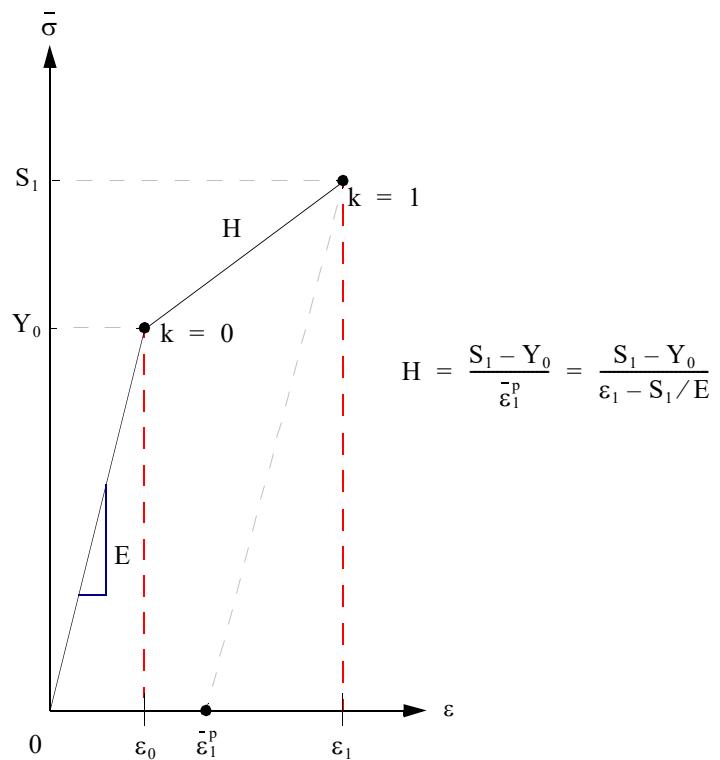


Figure 9-106 Single Workhardening Slope

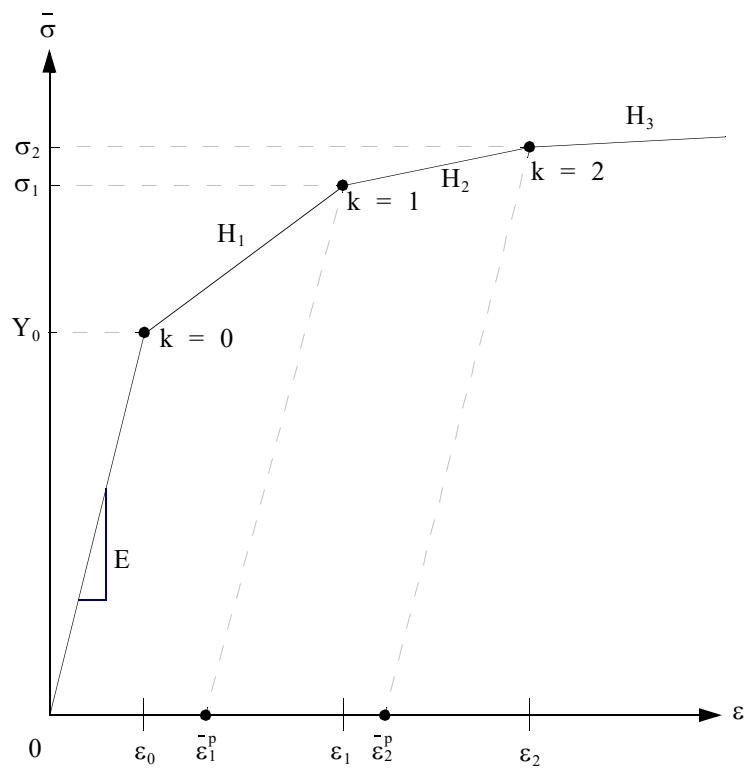


Figure 9-107 Stress vs Total Strain TABLES1(TYPE1)

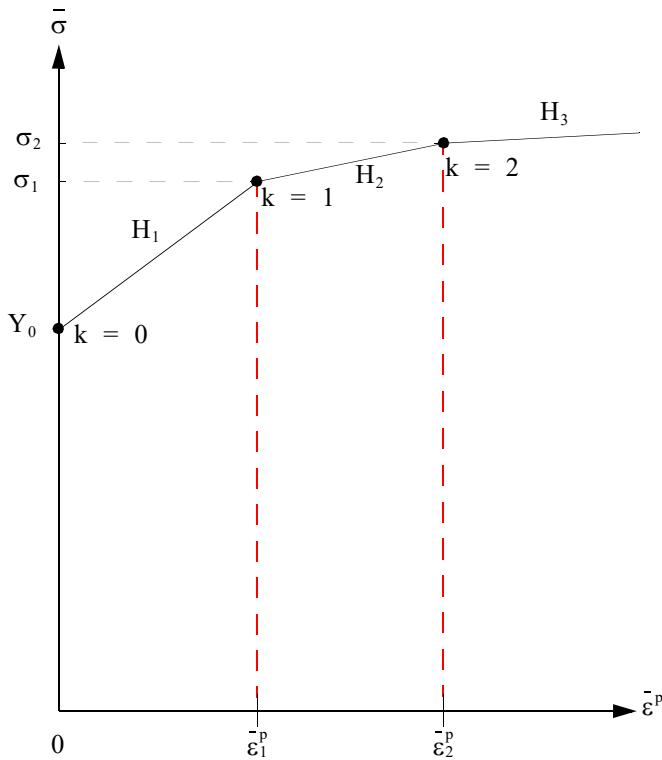


Figure 9-108 Stress vs Plastic Strain TABLES1(TYPE2) or TABL3D

If FORM = PERFECT,  $Y_0$  is the yield stress and H and FID should be blank. This is known as elastic-perfectly plastic behavior. If temperature-dependent yield is present, then FORM = TABLE should be used and the temperature-dependent yield entered through TABL3D.

3. For temperature-dependent materials, one can either define the temperature variation through the TABL3D option or include the MATTEP option. The first procedure is recommended.
4. In a large displacement analysis (LGDISP = 1, 2, 11, 12), the Cauchy stress and logarithmic strain are used.
5. The plastic deformation starts when the effective stress ( $\bar{\sigma}$ ) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ( $\bar{\epsilon}^p$ ). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

6. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[ 1 + \left( \frac{\dot{\varepsilon}}{C} \right)^{1/p} \right]$$

7. Hill's anisotropic model introduces orthotropic plastic material. The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_y - \sigma_z)^2 + G(\sigma_z - \sigma_x)^2 + H(\sigma_x - \sigma_y)^2 + 2L\tau_{yz}^2 + 2M\tau_{zx}^2 + 2N\tau_{xy}^2}$$

in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2} \left( \frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left( \frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left( \frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$R_{11} = \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the tensile yield stresses measured in material directions x, y and z, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{\ln(w_0/w)}{\ln(t_0/t)} = \frac{H + (2N - F - G - 4H)\sin^2\theta \cos^2\theta}{F \sin^2\theta + G \cos^2\theta}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\theta$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation,  $F$ ,  $G$ ,  $H$  and  $N$  can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left( r_{45} + \frac{1}{2} \right) \left( 1 + \frac{r_0}{r_{90}} \right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$aI_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters  $\alpha$  and  $\bar{\sigma}$  (effective stress coinciding with the yield stress) can be related to material constants  $c$  (cohesion) and  $\phi$  (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin \phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The keywords may appear in any order. However, aniso and press, are mutually exclusive, and cannot coexist.
10. All the alphanumeric fields are recognizable by the first four letters.
11. In SOL 400, MATEP is only supported for nonlinear elements with property extensions. This implies that for such elements, PBARL, PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral). If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS, SPROPMAP,-1 in the bulk data. See remark 9. under the NLMOPTS
12. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE and JHCOOK are only applicable to isotropic materials (MAT1).

**MATEP****Elasto-Plastic Material Properties - SOL 600**

Specifies elasto-plastic material properties to be used for large deformation analysis. Used in SOL 600 only.

**Format (Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered):**

| 1     | 2          | 3                      | 4        | 5         | 6         | 7                     | 8         | 9     | 10            |
|-------|------------|------------------------|----------|-----------|-----------|-----------------------|-----------|-------|---------------|
| MATEP | MID        | FORM                   | Y0       | FID       | RYIELD    | WKHARD                | METHOD    | H     |               |
|       | “Reffect”  | OPTION                 | RTID     | C         | P         |                       |           |       |               |
|       | “Aniso”    | N/A                    | R11 or M | R22 or C1 | R33 or C2 | R12 or C3             | R23 or C6 | R31   | (Hill/Barlat) |
|       | “ORNL”     | OPTION                 | Yc10     | TID       | N/A       |                       |           |       |               |
|       | “Press”    | OPTION                 | ALPHA    | BETA      | CRACKS    | SOFTEN                | CRUSHES   | SRFAC |               |
|       | “Gurson”   | Q1                     | Q2       | INITIAL   | CRITICAL  | FAILURE               | NUCL      | MEAN  |               |
|       |            | SDEV                   | NFRAC    |           |           |                       |           |       |               |
|       | “Chaboche” | R0                     | Rinf     | B         | C         | Gam                   | Kap       | N     |               |
|       |            | Qm                     | $\mu$    | $\eta$    |           |                       |           |       |               |
|       | “PwrLaw”   | A                      | M        | B         | N         | $\sigma_0 \epsilon_0$ |           |       |               |
|       | “Kumar”    | B0                     | A        | B1        | B2        | B3                    | T(B0)     | T(A)  |               |
|       |            | N                      | B4       | B5        | B6        | T(N)                  |           |       |               |
|       | “JhCook”   | A                      | B        | N         | C         | M                     | Tmelt     | Troom |               |
|       |            | $\dot{\epsilon}_0$ DOT |          |           |           |                       |           |       |               |
|       | “YldOpt”   | OPTION                 |          |           |           |                       |           |       |               |
|       | “Units”    | Uopt                   |          |           |           |                       |           |       |               |
|       | “VParam”   | Nvp                    | Vp1-1    | Vp2-1     | Vp1-2     | Vp2-2                 |           |       |               |
|       | “ImpCreep” | VMISES                 |          |           |           |                       |           |       |               |

**Example:**

|       |       |       |     |     |  |      |        |  |  |
|-------|-------|-------|-----|-----|--|------|--------|--|--|
| MATEP | 100   | table |     | 20  |  | HILL | ADDRAD |  |  |
|       | ANISO |       | 1.1 | 0.9 |  | 1.02 |        |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of MAT1, MAT2, MATORT or MAT9 entry. (Integer > 0).            |
| FORM       | Selects a form of stress-plastic strain function to be specified (Character):        |
|            | SLOPE for defining non-zero H. (Default)   |
|            | TABLE for defining the function in TABLES1 or TABL3Di units of table are unspecified |
|            | PERFECT for defining perfectly plastic material. (H is zero or blank)                |

| Descriptor | Meaning  |
|------------|--|
| Y0         | Initial yield stress $Y_0$ or hydrostatic stress for Mohr-Coulomb materials. See Remark 2.<br>(Real > 0 or blank)  |
| FID        | Identification number of TABLES1 or TABL3Di entry. (Integer > 0 or blank)  |
| RYIELD     | Enter one of the following yield criteria rules. (Character, Default = vMises if there are no continuation lines. Otherwise, the default will be adjusted to agree with information on the continuation lines. For example, if the second line has PRESS in field 2 and LIN in field 3, the default for RYIELD will be LinMohr.) |
| ELASTIC    | for purely elastic material (implies von Mises for multi-directional stresses). Vmises for von Mises yield criteria.   |
| VMISES     | von Mises yield criteria   |
| HILL       | for Hill's 1948 yield criteria.  |
| BARLAT     | for Barlat's 1991 yield criteria   |
| LINMOHR    | for Linear Mohr-Coulomb yield criteria. (See SOL 600 Notes/ Remarks regarding this entry in combination with MATS1. This entry overrides.)   |
| PBLMOHR    | for parabolic Mohr-Coulomb yield criteria. (See SOL 600 Notes/Remarks regarding this entry in combination with MATS1. This entry overrides.)   |
| BUYUK      | for the Buyukozturk concrete model.  |
| ORNL       | for one of the ORNL types. (see "Ornl" below)  |
| GENPLAS    | for Generalized Plasticity model   |
| VISCPLAS   | for Viscoplastic model using user subroutine UVSCPL  |
| RIGID      | for Rigid-Plastic material (no elasticity, von Mises yield)  |
| IMPCREEP   | for implicit creep model combining both plasticity and creep, von Mises yield criteria.  |
| WKHARD     | Selects a hardening rule defined by various work-hardening rules (Character):  |
| ISOTROP    | for isotropic hardening. (Default)   |
| KINEM      | for kinematic hardening.   |
| COMBINE    | for combination between kinematic and isotropic hardening  |
| CHABOCHE   | for Chaboche formulation - like combined (see Marc Vol C documentation for details for this model)   |
| PWRLAW     | for Power Law formulation (see Marc Vol C documentation for details of this model, for superplastic forming, this is the only available option)  |

| Descriptor | Meaning   |
|------------|---|
| RPWRLAW    | for Rate Power Law formulation (see Marc Vol C documentation for details of this model)   |
| JHCOOK     | for Johnson-Cook formulation (see Marc Vol C documentation for details of this model)   |
| KUMAR      | for Kumar formulation - (see Marc Vol C documentation for details of this model). See Remark 4.   |
| METHOD     | Selects a material processing method. See Remark 14. (Character):   |
| ADDMEAN    | for large strain additive decomposition using the mean normal process. (Default)  |
| ADDRAD     | for large strain additive decomposition using the radial return process. (SOL 600 only)   |
| MULTRAD    | for multiplicative decomposition (FeFp) using the radial return process   |
| MULTPLN    | for plane stress or strain FeFp multiplicative decomposition. (SOL 600)   |
| SMLMEAN    | for small strain additive decomposition using the mean normal process. (SOL 600 only)   |
| SMLRAD     | for small strain additive decomposition using the radial return process. (SOL 600 only)   |
| H          | plasticity modulus, ignored if FID field is specified. H has the same meaning as H in Remark 2 of the MATS1 entry. (Real $\geq 0$ ; Default = 0.0)  |
| “Reffect”  | A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter if rate-dependent effects are not to be included).  |
| OPTION     | Selects an option for strain-rate dependent yield stress (Character):   |
| TABLE      | for TABLES1 input. (Default)  |
| COWPER     | for Cowper and Symonds model. See Remark 6.   |
| RTID       | TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate. |
| C          | Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)  |
| P          | Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)  |
| “Aniso”    | A keyword signifying that the following data ( $R_{ij}$ ) pertain to the Hill’s ( $R_{ij}$ ) or Barlat’s ( $M, C_i$ ) yield criteria. See Remark 9. and Marc Vol A and C documentation (enter only RYIELD = HILL or BARLAT).  |
| Rij        | Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real $> 0$ ; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill), or  |

| Descriptor | Meaning  |
|------------|--|
| M          | Barlat M coefficient (see Marc Vol A and C documentation)  |
| C1         | Barlat C1 coefficient (see Marc Vol A and C documentation)   |
| C2         | Barlat C2 coefficient (see Marc Vol A and C documentation)   |
| C3         | Barlat C3 coefficient (see Marc Vol A and C documentation)   |
| C6         | Barlat C6 coefficient (see Marc Vol A and C documentation)   |
| "ORNL"     | A keyword signifying that the following data pertains to the ORNL's yield criteria. Only enter if RYIELD = ORNL. See Remark 7.   |
| OPTION     | Selects an option for ORNL yield criteria (Character):<br>NORM for normal ORNL model for stainless steel. (Default)<br>CRMO for ORNL 2-1/4 Cr-Mo steel model.<br>REVP for ORNL reversed plasticity model.<br>ARST for ORNL full alpha reset model. |
| Yc10       | Equivalent 10th cycle tensile yield stress. (Real > 0)   |
| TID        | Identification number of TABLES1 entry for normalized 10th cycle stress-plastic strain curve. (Integer; Default = 0)   |
| "Press"    | A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield=LinMohr, PblMohr or concrete cracking is to be simulated. See Remarks 8. and 10.   |
| OPTION     | Selects an option for pressure-dependent yield criteria. (Character):<br>LIN for linear Mohr-Coulomb model. (Default)<br>PBL for parabolic Mohr-Coulomb model.<br>CONC for Buyukozturk concrete model.   |
| ALPHA      | Specifies a parameter alpha for linear Mohr-Coulomb model or concrete model. (Real > 0 or blank)   |
| BETA       | Specifies a parameter beta for parabolic Mohr-Coulomb model or Buyukozturk concrete model (not used for linear Mohr-Coulomb model). (Real > 0 or blank)  |
| CRACKS     | Critical cracking stress for Buyukozturk concrete model. (Real > 0 or blank)   |
| SOFTEN     | Tension-softening modulus for Buyukozturk concrete model. (Real $\geq$ 0; Default = 0)   |
| CRUSHS     | Strain at which material crushes. (Real > 0; Default = 1.E10)  |
| SRFAC      | Shear retention factor defining shear stress carrying capacity when crack closes for Buyukozturk concrete model. ( $0 \leq$ Real < 1; Default = 0)   |
| "Gurson"   | A keyword signifying that the following data pertain to the modified Gurson model for porous metal plasticity with damage effects. Enter only if the Gurson damage model is to be used. See Remark 9.  |
| Q1         | First coefficient for the Gurson yield function. (Real $\geq$ 0; Default = 1.5)  |

| Descriptor | Meaning   |      |                    |        |   |        |                                   |
|------------|---|------|--------------------|--------|---|--------|-----------------------------------|
| Q2         | Second coefficient for the Gurson yield function. (Real $\geq 0$ ; Default = 1)   |      |                    |        |   |        |                                   |
| INITIAL    | Initial void volume fraction. ( $0 \leq \text{Real} < 1$ ; Default = 0)   |      |                    |        |   |        |                                   |
| CRITICAL   | Critical void volume fraction at which void coalescence starts. ( $0 < \text{Real} < 1$ ; Default = 0.2)  |      |                    |        |   |        |                                   |
| FAILURE    | Failure void volume fraction at which the material loses strength. ( $1./q_1 < \text{Real} < 1$ ; Default = 0.733)  |      |                    |        |   |        |                                   |
| NUEL       | Select a method of void nucleation (character, Default = strain): <table> <tr> <td>NONE</td><td>for no nucleation.</td></tr> <tr> <td>STRAIN</td><td>for plastic strain controlled nucleation.</td></tr> <tr> <td>STRESS</td><td>for stress controlled nucleation.</td></tr> </table> | NONE | for no nucleation. | STRAIN | for plastic strain controlled nucleation. | STRESS | for stress controlled nucleation. |
| NONE       | for no nucleation.  |      |                    |        |   |        |                                   |
| STRAIN     | for plastic strain controlled nucleation.   |      |                    |        |   |        |                                   |
| STRESS     | for stress controlled nucleation.   |      |                    |        |   |        |                                   |
|            | See Remark 9.   |      |                    |        |   |        |                                   |
| MEAN       | Mean strain or stress for void nucleation. (Real $> 0$ ; Default = 0.3)   |      |                    |        |   |        |                                   |
| SDEV       | Standard deviation in the assumed normal distribution of nucleation strain or stress. (Real $> 0$ ; Default = 0.01)   |      |                    |        |   |        |                                   |
| NFRAC      | Volume fraction of nucleating particles for void nucleation. ( $0 < \text{Real} < 0.5$ ; Default = 0.04)  |      |                    |        |   |        |                                   |
| “Chaboche” | A keyword specifying the following data pertains to the Chaboche model (Enter only if WKHARD=Chaboche)  |      |                    |        |   |        |                                   |
| R0         | R0 for isotropic hardening. (Real)  |      |                    |        |   |        |                                   |
| Rinf       | Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization). (Real)   |      |                    |        |   |        |                                   |
| B          | b coefficient for isotropic hardening. (Real)   |      |                    |        |   |        |                                   |
| C          | C coefficient for kinematic hardening (Real)  |      |                    |        |   |        |                                   |
| Gam        | Gamma coefficient for kinematic hardening (Real)  |      |                    |        |   |        |                                   |
| Kap        | Kappa value for viscosity model. (Real)   |      |                    |        |   |        |                                   |
| N          | n coefficient for viscosity model. (Real)   |      |                    |        |   |        |                                   |
| Qm         | Qm coefficient for isotropic hardening. (Real)  |      |                    |        |   |        |                                   |
| $\mu$      | coefficient for isotropic hardening. (Real)   |      |                    |        |   |        |                                   |
| $\eta$     | coefficient to introduce progressive memory. (Real)   |      |                    |        |   |        |                                   |
| “PwrLaw”   | A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C, ISOTROPIC option for more details. Enter only if WKHARD = PWRLAW or RPWRLAW).  |      |                    |        |   |        |                                   |
| A          | Coefficient A. (Real)   |      |                    |        |   |        |                                   |
| M          | Coefficient m. (Real)   |      |                    |        |   |        |                                   |
| B          | Coefficient B. (Real)   |      |                    |        |   |        |                                   |

| Descriptor               | Meaning  |
|--------------------------|--|
| N                        | Exponent n. (Real)   |
| $\sigma_0 \varepsilon_0$ | Enter $\sigma_0$ for rate power law or $\varepsilon_0$ for the power law model (Real) See Marc Vol C, ISOTROPIC option if this field is left blank or set to 0.0 |
| "Kumar"                  | A keyword specifying the following data pertains to the Kumar model (see Marc Vol C , ISOTROPIC option for more details. Enter only if Wkhard =Kumar).           |
| B0                       | Coefficient B0. (Real)   |
| A                        | Coefficient A. (Real) If A=0.0, B1,B2, and B3 are used   |
| B1                       | Coefficient B1. (Real)   |
| B2                       | Coefficient B2. (Real)   |
| B3                       | Coefficient B3. (Real)   |
| T(B0)                    | TABLEM1 ID for coefficient B0. (Integer)   |
| T(A)                     | TABLEM1 ID for coefficient A. (Integer)  |
| N                        | Exponent n. (Real) If N=0.0, B4, B5 and B6 are used  |
| B4                       | Coefficient B4. (Real)   |
| B5                       | Coefficient B5. (Real)   |
| B6                       | Coefficient B6. (Real)   |
| T(N)                     | TABLEM1 ID for exponent n  |
| "JhCook"                 | A keyword specifying the following data pertains to the Johnson-Cook model (see Marc Vol C , ISOTROPIC option for more details. Enter only if WKHARD =JHCOOK).   |
| A                        | Coefficient A. (Real)  |
| B                        | Coefficient B. (Real)  |
| N                        | Exponent n. (Real)   |
| C                        | Coefficient C. (Real)  |
| M                        | Exponent m. (Real)   |
| Tmelt                    | Melting temperature. (Real)  |
| Troom                    | Ambient temperature. (Real)  |
| $\dot{\varepsilon}_0$    | Reference strain rate. (Real)  |
| "YldOpt"                 | A keyword specifying one of the following yield models will be used.   |
| OPTION                   | Select a yield option from the list below:   |
| GENPL                    | General plasticity viscoplastic model using user subroutine UVSCPL   |
| RIGID                    | Rigid-plastic material, no elasticity, vonMises yield  |
| IMCREEP                  | Implicit creep model, both plasticity and creep with vonMises yield criteria   |

| Descriptor | Meaning  |
|------------|--|
| "Units"    | A keyword specifying if data is read from a database and if so, the units to be used if data is read from a database. (Enter only if necessary). |
| 4          | Read from Marc database for flow stress (default units)  |
| 5          | Use MATILDA database (default units)   |
| 6          | Data read from input file in SI-mm units   |
| 7          | Data read from input file in SI-m units  |
| 8          | Data read from input file in US units  |
| "VParam"   | A keyword specifying that the number of viscoplastic parameters. Up to two viscoelastic parameters are to be entered (used when RYIELD=ViscPlas) |
| Nvp        | Number of viscoplastic parameters. (Integer must be 0, 1, or 2)  |
| Vp1-1      | First Viscoplastic parameter for Nvp=1. (Real; Default = 0.0)  |
| Vp2-1      | Second viscoplastic parameter for Nvp=1. (Real; Default = 0.0)   |
| Vp1-2      | First Viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)   |
| Vp2-2      | Second viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)  |
| "ImpCreep" | A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used when RYIELD=ImpCreep).         |
| VMISES     | Equivalent (von Mises) tensile yield stress. Overrides any similar entry made for this material elsewhere. (Real, no Default)                    |

**Remarks:**

- Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
- If  $Y_0$  is not specified, FID field referring to a stress-plastic strain curve must be provided. If  $Y_0$  is specified without FID field, the material is assumed perfectly plastic or has one slope, H. If both  $Y_0$  and FID fields are specified, FID supersedes the  $Y_0$  field and the first data point in TABLES1 represents  $Y_0$ .

The initial yield point corresponds to the first data point in the function specified on the FID field.

$$Y(\bar{\varepsilon}^p) = Y_0(1 + b \bar{\varepsilon}^p)^n$$

where  $Y_0$  is an initial yield stress,  $b$  and  $n$  are parameters characterizing the stress-strain relationship.

In case of an anisotropic material, the initial yield point corresponds to the reference yield stress ( $Y_a$  in Remark 7.).

- For SOL 600, the form of the stress-strain curve is determined by parameters MRTABLS1 and MRTABLS2.

4. The plastic deformation starts when the effective stress ( $\bar{\sigma}$ ) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ( $\dot{\varepsilon}^p$ ). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

with

$$\bar{\sigma} = \sqrt{\frac{1}{2} [(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

$$\begin{aligned} R_{11} &= \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a} \\ R_{12} &= \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a} \end{aligned}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\alpha$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation,  $F$ ,  $G$ ,  $H$  and  $N$  can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2}\right) \left(1 + \frac{r_0}{r_{90}}\right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

5. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[ 1 + \left( \frac{\dot{\varepsilon}}{C} \right)^{1/p} \right]$$

6. Hill's anisotropic model introduces orthotropic plastic material. This option can only be combined with orthotropic or anisotropic elastic material (i.e., with MAT2, MATORT or MAT9). The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2}$$

in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2} \left( \frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left( \frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left( \frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$\begin{aligned} R_{11} &= \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a} \\ R_{12} &= \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a} \end{aligned}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\alpha$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation,  $F$ ,  $G$ ,  $H$  and  $N$  can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0, \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2}\right) \left(1 + \frac{r_0}{r_{90}}\right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

7. The elasticity constants must be isotropic for ORNL plasticity except for normal ORNL model for stainless steel. The 10th cycle stress-plastic strain curve in TID field should be a normalized function so that the yield stress at zero plastic strain is unity.
8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$\alpha I_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters  $\alpha$  and  $\bar{\sigma}$  (effective stress coinciding with the yield stress) can be related to material constants  $c$  (cohesion) and  $\phi$  (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin\phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The Drucker-Prager plasticity models can only be combined with isotropic elasticity. The Buyukozturk concrete plasticity model is a particular form of the generalized Drucker-Prager plasticity model, which is developed specifically for plane stress cases by Buyukozturk. The Buyukozturk yield function is expressed as

$$\beta\sqrt{3}Y I_1 + \gamma I_1^2 + 3J_2 - Y^2 = 0$$

where  $\beta$  is a user-specified constant,  $\gamma$  is an internal parameter (set to 0.2) with no user's access, and  $Y$  is the yield stress.

The Buyukozturk concrete plasticity model is coupled with crack and crush capability, which is designed for a low-tension material. The low-tension material develops a crack in the perpendicular direction to the maximum principal stress when it exceeds a critical value. The tension softening modulus can be specified (in absolute value) by the user to process the cracking process gradually. The default value (0.) is intended for a sudden cracking with a complete loss of the stiffness upon cracking. After the initial crack, a second crack can initiate in the perpendicular direction to the first crack. Likewise, the third crack can be formed in 3D solid elements. The loading may reverse the direction after the crack is formed. In this case, the crack will close and some load carrying capacity is resumed. The compression capability is fully resumed and the shear stresses may be transmitted over the crack surface with a reduced stiffness by a factor specified as shear retention factor. The material may fail in compression by crushing. The input value for the crush strain is positive, which implies an absolute value of a compressive strain. The material loses its integrity for good upon crushing. The reinforcement bars may be simulated by adding REBAR elements.

10. The Gurson model for porous metal plasticity may be used only with isotropic hardening rule. All other hardening rules will be ignored if Gurson model is selected. The Gurson model modified by Tvergaard and Needleman is designed for porous metal plasticity with damage effects in the ductile material. The material is assumed to form voids under loading, which grow, coalesce, then leads to crack formation and eventually failure. This process is a function of hydrostatic stress and the void volume fraction  $f_v$ . The yield function is established as follows:

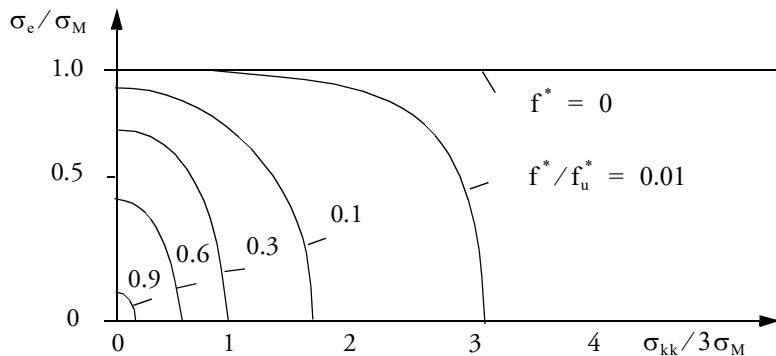
$$\frac{3J_2}{Y^2} + 2q_1 f_v^* \cosh\left(\frac{q_2 I_1}{2Y}\right) - [1 - (q_1 f_v^*)^2] = 0$$

in which  $Y$  denotes a yield stress of the fully dense matrix material,  $I_1$  denotes the first invariant of stresses, and the modified void volume fraction  $f_v^*$  is computed by

$$f_v^* = f_v \quad \text{if } f_v \leq f_v^c$$

$$f_v^* = f_v^c + \left( \frac{f_v^u - f_v^c}{f_v^f - f_v^c} \right) (f_v - f_v^c) \quad \text{if } f_v > f_v^c$$

where  $f_v^c$  is the critical void volume fraction,  $f_v^f$  is the void volume at failure and  $f_v^u = 1/q_1$ . The solid loses all stress carrying capability when the void volume fraction reaches  $f_v^f$ .



11. The evolution of damage as measured by void volume fraction is due to void nucleation and void growth. Void nucleation occurs by debonding of the second phase particles. The strain for nucleation depends on the particle sizes. Assuming a normal distribution of particle sizes, the void nucleation itself is modeled as a normal distribution in strains if nucleation is strain-controlled. If the void nucleation is assumed to be stress controlled in the matrix, a normal distribution is assumed in stresses. The void volume fraction changes due to the growth of existing voids and nucleation of new voids, i.e.,

$$\dot{f}_v = \dot{f}_{\text{growth}} + \dot{f}_{\text{nucleation}}$$

in which the void growth can be determined based on the compressibility of the material

$$\dot{f}_{\text{growth}} = (1 - f_v) \dot{\epsilon}_{kk}^p$$

and the nucleation can be defined either as strain or stress-controlled with a normal distribution about the mean value. In case of strain-controlled nucleation, the rate is expressed as

$$\dot{f}_{\text{nucleation}} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[ -\frac{1}{2} \left( \frac{\bar{\epsilon}_m^p - \epsilon_n}{S} \right)^2 \right] \dot{\epsilon}_m^p$$

where  $f_v^n$  is the volume fraction of void forming particles,  $\bar{\epsilon}_m^p$  denotes the effective plastic strain in the matrix material, and the void nucleation strain is assumed normally distributed with a mean value of  $\epsilon_n$  and a standard deviation of  $S$ . In case of stress-controlled nucleation, the rate is expressed as

$$\dot{f}_{\text{nucleation}} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[ -\frac{1}{2} \left( \frac{\sigma + \frac{1}{3}\sigma_{kk} - \sigma_n}{S} \right)^2 \right] \left( \frac{\dot{\sigma}}{\sigma} + \frac{1}{3} \frac{\dot{\sigma}_{kk}}{\sigma} \right)$$

If the size of the second phase particles are widely dispersed, the standard deviation would be larger than more uniform cases. A typical values for an engineering alloy as suggested by numerical experiments are set as default values for  $\epsilon_n$ ,  $S$ , and  $f_v^n$ .

12. The keywords may appear in any order. However, aniso, ORNL, press, and Gurson are mutually exclusive, and cannot coexist.
13. All the alphanumeric fields are recognizable by the first four letters.
14. For SOL 600, METHOD (first line, field 8) corresponds to Marc's PLASTICITY options as follows. METHOD must be the same for all MATEP entries in the SOL 600 input but this is not enforced. The value of METHOD on MATEP with the smallest MID governs who the PLASTICITY option is set. If METHOD is left blank, it may alternatively be set by PARAM,MARCPPLAS.

| Name    | Marc PLASTICITY parameter value |
|---------|---------------------------------|
| Addmean | 3                               |
| Addrad  | 4                               |
| Multrad | 5                               |
| Multpln | 6                               |
| Smlmean | 1                               |
| Smlrad  | 2                               |

15. The Bulk Data fields denoted by N/A are different from blank fields. Those fields with N/A are not used currently, but the space is reserved in case of future additions. On the other hand, the blank fields which have no specifications are neither used nor reserved (any memory space) for future use. The Method field determines the options under Marc parameter PLASTICITY.

The initial yield stress should be extracted from the table provided in the FID field. The yield stress-plastic strain function specified in TABLES1 under FID field is not a normalized function.

The anisotropic material parameters Rij are equivalent to Marc input data as follows:

R11 = YRDIR1  
 R22 = YRDIR2  
 R33 = YRDIR3  
 R12 = YRSHR1  
 R23 = YRSHR2  
 R31 = YRSHR3

The crack/crush capability in Marc may be combined with other isotropic material options.

16. The strain effect on the yield stress (Reffect) is specified under the model definition option STRAIN RATE in Marc.
17. This Bulk Data entry accommodates Marc's input data under the model definition options ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, RATE EFFECTS, WORK HARD, DAMAGE and CRACK DATA as well as the parameter PLASTICITY.
18. The fields without default values can be left blank only if they are specifically permitted to be blank.
19. RYIELD equates to Marc's ISOTROPIC (3,2) field.
20. WKHARD equates to Marc's ISOTROPIC (3,3) field.

21. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE, PWRLAW, RPWRLAW, JHCOOK, and KUMAR are only applicable to isotropic materials (MAT1).

**MATEP****Elasto-Plastic Material - SOL 700**

Elasto-plastic material properties for SOL 700 only.

**Format:**

Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

| 1     | 2         | 3       | 4     | 5     | 6      | 7     | 8     | 9     | 10 |
|-------|-----------|---------|-------|-------|--------|-------|-------|-------|----|
| MATEP | MID       | FORM    | Y0    | FID   | RYIELD |       | TYPE  | H     |    |
|       | “Reffect” | OPTION1 | RTID  | C     | P      |       |       |       |    |
|       | “JhCook”  | A       | B     | N     | C      | M     | Tmelt | Troom |    |
|       |           | ε0DOT   | CP    |       |        |       |       |       |    |
|       | “PwrLaw”  | A       | M     | B     | N      | C     |       |       |    |
|       | “Poly”    | A       | B     | C     | D      | E     | F     | YMAX  |    |
|       | “TMmodel” | A       | B     | C     | D      | M     | εSM   | Tmelt |    |
|       |           | Troom   | YCR   | E     | K      | ε0DOT | CP    |       |    |
|       | “ZAmodel” | A       | B     | N     | C      | M     | ε0DOT | D     |    |
|       |           | CP      |       |       |        |       |       |       |    |
|       | “OrthoCr” | E       | NU    | RELV  | TYPE   | TIDXX | TIDYY | TIDZZ |    |
|       |           | TIDXY   | TIDYZ | TIDZX | TIDSР  |       |       |       |    |
|       | “Soil”    | TABLE   | TYPE  | VALUE | CUTOFF | A0    | A1    | A2    |    |
|       |           | YIELD   | YSTYP |       |        |       |       |       |    |

**Example:**

|       |           |        |       |     |        |  |  |       |  |
|-------|-----------|--------|-------|-----|--------|--|--|-------|--|
| MATEP | 7         | SLOPE  | 1.2E6 |     | VMISES |  |  | 1.0E3 |  |
|       | “Reffect” | COWPER |       | 2.0 | 4.0    |  |  |       |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of MAT1 or MATORT entry. MATORT is only acceptable for RYIELD=ORTHOCR and MAT1 is acceptable for all other yield criteria except ORTHOCR. (Integer > 0; required.) |
| FORM       | Selects a form of stress-plastic strain function to be specified for RYIELD=VMISES. (Character; default=SLOPE):  |
| SLOPE      | Defines non-zero H.  |
| TABLE      | Defines the function using TABLED1 entry. Units of table are unspecified.  |
| PERFECT    | Defines perfectly plastic material (H is zero or blank and FID is blank).  |

| Descriptor | Meaning  |
|------------|--|
| Y0         | Initial yield stress for RYIELD=VMISES and ORTHOCR. (Real > 0.0; default=blank)  |
| FID        | Identification number of TABLED1 for RYIELD=VMISES. (Integer > 0; default=blank)   |
| RYIELD     | Enter one of the following yield criteria rules. (Character; Default = VMISES)   |
| VMISES     | von Mises yield criteria. It supports “Reffect” option. See Remarks <a href="#">1.-8</a> .   |
| JHCOOK     | For Johnson-Cook formulation. It supports “JhCook” option. See Remarks <a href="#">9.</a> and <a href="#">10</a> .   |
| RPWRLAW    | Rate power law. It supports “PwrLaw” option. See Remark <a href="#">11</a> .   |
| POLY       | Polynomial Yield Model. It supports “Poly” option. See Remark <a href="#">12</a> .   |
| TMMODEL    | Tanimura-Mimura Yield Model. It supports “TMmodel” option. See Remark <a href="#">13</a> .   |
| ZAMODEL    | Zerilli-Armstrong Yield Model. It supports “ZAmode” option. See Remarks <a href="#">14</a> .   |
| ORTHOCR    | Orthotropic Crushable Model. It supports “OrthoCr” option. It is only available for solid elements with MATORT. See Remarks <a href="#">15.-18</a> .   |
| SOIL       | Elastoplastic Crushable Model. It supports “Soil” option. It is only available for solid elements. See Remarks <a href="#">19.-22</a> .  |
| TYPE       | The type of stress and strain defined in FID. (Character; default=TRUE)  |
| ENG        | Engineering stress and strain.   |
| TRUE       | True stress and strain.  |
| PLAST      | True stress and plastic strain.  |
| PMOD       | Plastic modulus and true stress.   |
| H          | Plasticity modulus. (Real > 0; Default = 0).   |
| “Reffect”  | A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABLED1 is entered and one of the independent variables is strain rate, this field and its options should not be specified. |
| OPTION1    | Selects an option for strain-rate dependent yield stress (Character; default=TABLE):   |
| TABLE      | Specifies that RTID input used to define rate dependency.  |
| COWPER     | Specifies the Cowper and Symonds rate-dependency model.  |
| RTID       | TABLED1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.                    |

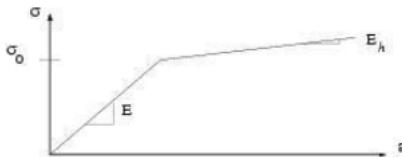
| Descriptor            | Meaning  |
|-----------------------|--|
| C                     | Specifies the constant C for Cowper and Symonds model. (Real; Default = 1.0)   |
| P                     | Specifies the constant P for Cowper and Symonds model. (Real; Default = 1.0)   |
| "Jhcook"              | Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature. |
| A                     | Static yield stress. (Real > 0.0; required.)   |
| B                     | Hardening parameter. (Real; default=0.0)   |
| N                     | Hardening exponent. (Real; default=1.0)  |
| C                     | Strain-rate parameter. (Real; default=0.0)   |
| M                     | Temperature exponent. (Real; default=1.0)  |
| TMELT                 | Melt temperature. (Real; default=1.E20)  |
| TROOM                 | Room temperature. (Real; default=293.0)  |
| $\dot{\epsilon}$ 0DOT | Reference strain rate. (Real > 0.0; default=1.0)   |
| CP                    | Specific heat. (Real; default=1.E20)   |
| "PwrLaw"              | Rate power law   |
| A                     | Hardening parameter. (Real; default=0.0)   |
| M                     | Hardening exponent. (Real; default=1.0)  |
| B                     | Initial yield parameter. (Real > 0.0; required)  |
| N                     | Strain rate exponent. (Real; default=1.0)  |
| C                     | Minimum yield stress. (Real; default=1.E20)  |
| "Poly"                | Polynomial Yield Model   |
| A                     | Initial yield parameter. (Real > 0.0; required)  |
| B                     | Coefficient B. (Real; default=0.0)   |
| C                     | Coefficient C. (Real; default=0.0)   |
| D                     | Coefficient D. (Real; default=0.0)   |
| E                     | Coefficient E. (Real; default=0.0)   |
| F                     | Coefficient F. (Real; default=0.0)   |
| YMAX                  | Maximum yield stress. (Real; default=1.E20)  |
| "SGmodel"             | Steinberg-Guinan Yield Model   |
| A1~A4                 | Yield parameters. (Real > 0.0; required.)  |
| H, B                  | Yield parameters. (Real; default=0.0)  |
| TMELT                 | Melt temperature. (Real; default=1.E20)  |
| TROOM                 | Room temperature. (Real; default=293.0)  |
| CP                    | Specific heat. (Real; default=1.E20)   |

| Descriptor       | Meaning  |  |       |                                  |          |                      |
|------------------|--|--|-------|----------------------------------|----------|----------------------|
| "TMmodel"        | Tanimura-Mimura Yield Model  |  |       |                                  |          |                      |
| A                | Static yield parameter. (Real > 0.0; Required)   |  |       |                                  |          |                      |
| B                | Hardening parameter. (Real; default=0.0)   |  |       |                                  |          |                      |
| C                | Strain rate parameter. (Real; default=0.0)   |  |       |                                  |          |                      |
| D                | Strain rate parameter. (Real; default=0.0)   |  |       |                                  |          |                      |
| M                | Temperature exponent. (Real; default=1.0)  |  |       |                                  |          |                      |
| $\epsilon_{SM}$  | Quasi-static strain rate. (Real > 0.0; default=1.0)  |  |       |                                  |          |                      |
| TMELT            | Melt temperature. (Real; default=1.E20)  |  |       |                                  |          |                      |
| TROOM            | Room temperature. (Real; default=293.0)  |  |       |                                  |          |                      |
| YCR              | Critical yield stress. (Real > 0.0; default=1.E20)   |  |       |                                  |          |                      |
| E                | Strain rate parameter. (Real; default=0.0)   |  |       |                                  |          |                      |
| K                | Strain rate exponent. (Real; default=1.0)  |  |       |                                  |          |                      |
| $\epsilon_0$ DOT | Reference strain rate. (Real > 0.0; default=1.0)   |  |       |                                  |          |                      |
| CP               | Specific heat. (Real; default=1.E20)   |  |       |                                  |          |                      |
| "ZAmodel"        | Zerilli-Armstrong Yield Model.   |  |       |                                  |          |                      |
| A                | Static yield parameter. (Real > 0.0; required)   |  |       |                                  |          |                      |
| B                | Hardening parameter. (Real > 0.0; default=0.0)   |  |       |                                  |          |                      |
| N                | Hardening exponent. (Real > 0.0; default=1.0)  |  |       |                                  |          |                      |
| C                | Strain rate parameter. (Real; default=0.0)   |  |       |                                  |          |                      |
| M                | Temperature exponent. (Real; default=1.0)  |  |       |                                  |          |                      |
| $\epsilon_0$ DOT | Reference strain rate. (Real > 0.0; default=1.0)   |  |       |                                  |          |                      |
| D                | Bcc parameter. (Real; default=0.0) <table style="margin-left: 20px;"> <tr> <td>= 0</td> <td>Fcc metal</td> </tr> <tr> <td><math>\neq 0</math></td> <td>Bcc metal</td> </tr> </table>   |  | = 0   | Fcc metal                        | $\neq 0$ | Bcc metal            |
| = 0              | Fcc metal  |  |       |                                  |          |                      |
| $\neq 0$         | Bcc metal  |  |       |                                  |          |                      |
| CP               | Specific heat. (Real; default=1.E20)   |  |       |                                  |          |                      |
| "OrthoCr"        | Orthotropic Crushable Model.   |  |       |                                  |          |                      |
| E                | Young's modulus for the fully compacted material. (Real > 0.0; required)   |  |       |                                  |          |                      |
| NU               | Poisson's ratio for the fully compacted material. (-1.0 < Real < 0.5; required)  |  |       |                                  |          |                      |
| RELV             | Relative volume at which the material is fully compacted. (0.0 < Real < 1.0; required)   |  |       |                                  |          |                      |
| TYPE             | The type of data defined as the x-value in TIDXX-TIDS. (Character; default=CRUSH) <table style="margin-left: 20px;"> <tr> <td>CRUSH</td> <td>Crush factor (1-relative volume)</td> </tr> <tr> <td>RELVOL</td> <td>Relative volume V/V0</td> </tr> </table> |  | CRUSH | Crush factor (1-relative volume) | RELVOL   | Relative volume V/V0 |
| CRUSH            | Crush factor (1-relative volume)   |  |       |                                  |          |                      |
| RELVOL           | Relative volume V/V0   |  |       |                                  |          |                      |

| Descriptor | Meaning  |     |   |     |   |     |                  |
|------------|--|-----|---|-----|---|-----|------------------|
| TIDXX      | TABLED1 ID defining the variation of the (local) xx-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDYY      | TABLED1 ID defining the variation of the (local) yy-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDZZ      | TABLED1 ID defining the variation of the (local) zz-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDXY      | TABLED1 ID defining the variation of the (local) xy-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDYZ      | TABLED1 ID defining the variation of the (local) yz-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDZX      | TABLED1 ID defining the variation of the (local) zx-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)  |     |   |     |   |     |                  |
| TIDSR      | TABLED1 ID defining the variation of a yield factor (y-value) with the deviatoric strain rate (x-value). (Integer > 0; default=not used)   |     |   |     |   |     |                  |
| “Soil”     | Elastoplastic Crushable Model.   |     |   |     |   |     |                  |
| TABLE      | TABLED1 ID defining the variation of pressure (y-value) with crush factor or volumetric strain (x- value). (Integer > 0; required)   |     |   |     |   |     |                  |
| TYPE       | The type of data defined as the x-value in the table. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>Crush factor. (= 1-relative volume)</td></tr> <tr> <td>2.0</td><td>Volumetric (true) strain</td></tr> </table>                                  | 1.0 | Crush factor. (= 1-relative volume)                     | 2.0 | Volumetric (true) strain                                |     |                  |
| 1.0        | Crush factor. (= 1-relative volume)  |     |   |     |   |     |                  |
| 2.0        | Volumetric (true) strain   |     |   |     |   |     |                  |
| VALUE      | The value for cut-off stress. (Real < 0.0; See Remark 20.)   |     |   |     |   |     |                  |
| CUTOFF     | Cut-off stress. See Remark 21. (Real > 0.0; default=2.0) <table> <tr> <td>1.0</td><td>Pressure for total tensile failure</td></tr> <tr> <td>2.0</td><td>Pressure for tensile failure</td></tr> <tr> <td>3.0</td><td>Minimum pressure</td></tr> </table>      | 1.0 | Pressure for total tensile failure                      | 2.0 | Pressure for tensile failure                            | 3.0 | Minimum pressure |
| 1.0        | Pressure for total tensile failure   |     |   |     |   |     |                  |
| 2.0        | Pressure for tensile failure   |     |   |     |   |     |                  |
| 3.0        | Minimum pressure   |     |   |     |   |     |                  |
| A0, A1, A2 | Yield function constants. (Real; default=0.0)  |     |   |     |   |     |                  |
| YIELD      | Surface description. See Remark 22. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>The yield surface is defined as a function of p and J2.</td></tr> <tr> <td>2.0</td><td>The yield surface is defined as a function of p and sy.</td></tr> </table> | 1.0 | The yield surface is defined as a function of p and J2. | 2.0 | The yield surface is defined as a function of p and sy. |     |                  |
| 1.0        | The yield surface is defined as a function of p and J2.  |     |   |     |   |     |                  |
| 2.0        | The yield surface is defined as a function of p and sy.  |     |   |     |   |     |                  |
| YSTYP      | Type of YSURF Yield Surface description. See Remark 22. (Real > 0.0; default=1.0) <table> <tr> <td>1.0</td><td>DYNA definition</td></tr> <tr> <td>2.0</td><td>Dytran additional definition</td></tr> </table>  | 1.0 | DYNA definition   | 2.0 | Dytran additional definition                            |     |                  |
| 1.0        | DYNA definition  |     |   |     |   |     |                  |
| 2.0        | Dytran additional definition   |     |   |     |   |     |                  |

**Remarks:**

1. A bilinear stress-strain characteristic is used by specifying FORM=SLOPE, Y0 and H:



where the equivalent stress is given by  $\sigma = \sigma_0 + \frac{E \cdot E_h}{E - E_h} \varepsilon_p$

where

$\sigma_0$  = yield stress specified in the Y0 field

$E$  = Young's modulus

$E_h$  = hardening modulus specified in the H field

$\varepsilon_p$  = equivalent plastic strain

$\sigma$  = equivalent stress

2. A piecewise linear, stress-strain characteristic is used by specifying FID and TYPE (beams and shells only)

$$\sigma = [(\sigma_i - \sigma_{i-1})(\varepsilon - \varepsilon_{i-1}) / (\varepsilon_i - \varepsilon_{i-1})] + \sigma_{i-1}$$

The stress-strain characteristic used internally in Nastran SOL700 is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

|                         |               |
|-------------------------|---------------|
| True stress/true strain | (TYPE = TRUE) |
|-------------------------|---------------|

|                                       |              |
|---------------------------------------|--------------|
| Engineering stress/engineering strain | (TYPE = ENG) |
|---------------------------------------|--------------|

|                            |                |
|----------------------------|----------------|
| True stress/plastic strain | (TYPE = PLAST) |
|----------------------------|----------------|

|                             |               |
|-----------------------------|---------------|
| Plastic modulus/true stress | (TYPE = PMOD) |
|-----------------------------|---------------|

3. With solid elements, only an elastic perfectly plastic yield model is currently used. Only the Y0 field is used.
4. If FID is blank or zero, a bilinear stress-strain curve is assumed. If FID has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If FID is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If RTID is blank or zero and C and P are blank or zero, the yield stress does not vary with strain rate. If RTID has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. (C and P must be blank or zero.)

If RTID is blank or zero and C and P are defined, the enhancement of the yield stress with strain rate is calculated as

$$\frac{\sigma_d}{\sigma_y} = I + \left( \frac{\dot{\varepsilon}_p}{C} \right)^{1/p}$$

where  $\sigma_d$  is the dynamic stress,  $\sigma_y$  is the static yield stress, and  $\dot{\varepsilon}_p$  is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined, it overrides the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see FID and RTID fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. The Johnson Cook yield stress is computed from

$$\sigma_y = (A + B\varepsilon_p^N) \left[ 1 + C \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right) \right] (1 - T^{*M})$$

where

|                 |                            |
|-----------------|----------------------------|
| $\varepsilon_p$ | = effective plastic strain |
|-----------------|----------------------------|

|       |                                     |
|-------|-------------------------------------|
| $T^*$ | $= \frac{T - TROOM}{TMELT - TROOM}$ |
|-------|-------------------------------------|

|                     |                         |
|---------------------|-------------------------|
| $\dot{\varepsilon}$ | = reference strain rate |
|---------------------|-------------------------|

|                       |                         |
|-----------------------|-------------------------|
| $\dot{\varepsilon}_0$ | = reference strain rate |
|-----------------------|-------------------------|

|   |               |
|---|---------------|
| T | = temperature |
|---|---------------|

|       |                    |
|-------|--------------------|
| TROOM | = room temperature |
|-------|--------------------|

|       |                    |
|-------|--------------------|
| TMELT | = melt temperature |
|-------|--------------------|

A, B, N, C, and M are input constants.

10. The reference strain rate for Johnson Cook yield model is per unit time.

11. The rate power law yield stress is computed from

$$\sigma_y = \text{MAX}(C, B + A\varepsilon_p^M \dot{\varepsilon}^N)$$

where

|                 |                            |
|-----------------|----------------------------|
| $\varepsilon_p$ | = effective plastic strain |
|-----------------|----------------------------|

|                     |                         |
|---------------------|-------------------------|
| $\dot{\varepsilon}$ | = reference strain rate |
|---------------------|-------------------------|

A, B, M,N and C are constants.

12. The polynomial yield stress is computed from

$$\sigma_y = \text{MIN}(\text{YMAX}, A + B\epsilon_p + C\epsilon_p^2 + D\epsilon_p^3 + E\epsilon_p^4 + F\epsilon_p^5)$$

where

|              |                            |
|--------------|----------------------------|
| $\epsilon_p$ | = effective plastic strain |
| YMAX         | = maximum yield stress     |

A, B, C,D,E and F are constants.

13. The Tanimura-Mimura yield stress is computed from

$$\sigma_y = \left[ A + B\epsilon_p + (C + D\epsilon_p) \left( 1 - \frac{A + B\epsilon_p}{\sigma_{cr}} \right) \ln \left( \frac{\dot{\epsilon}}{\dot{\epsilon}_s} \right) \right] (1 - T^{*M}) + E \left( \frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right)^K$$

where

|                    |                                     |
|--------------------|-------------------------------------|
| $\epsilon_p$       | = effective plastic strain          |
| $\sigma_{cr}$      | = critical yield stress             |
| $\dot{\epsilon}$   | = effective strain rate             |
| $\dot{\epsilon}_s$ | = quasi-static strain rate          |
| $\dot{\epsilon}_0$ | = reference strain rate             |
| $T^*$              | $= \frac{T - TROOM}{TMELT - TROOM}$ |
| $T$                | = temperature                       |
| $TROOM$            | = room temperature                  |
| $TMELT$            | = melt temperature                  |

A, B, D, M, E and K are constants

14. The Zerilli-Armstrong yield stress is computed from

$$\sigma_y = \begin{cases} (A + B\epsilon_p^N) e^{\left[ -MT + CT \ln \left( \frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right) \right]} & \text{for Fcc metals} \\ (A + B\epsilon_p^N) + De^{\left[ -MT + CT \ln \left( \frac{\dot{\epsilon}}{\dot{\epsilon}_0} \right) \right]} & \text{for Bcc metals} \end{cases}$$

where

|                 |                            |
|-----------------|----------------------------|
| $\varepsilon_p$ | = effective plastic strain |
|-----------------|----------------------------|

|                     |                         |
|---------------------|-------------------------|
| $\dot{\varepsilon}$ | = effective strain rate |
|---------------------|-------------------------|

|                       |                         |
|-----------------------|-------------------------|
| $\dot{\varepsilon}_0$ | = reference strain rate |
|-----------------------|-------------------------|

|     |               |
|-----|---------------|
| $T$ | = temperature |
|-----|---------------|

A, B, N,C,M and D are constants.

15. If the initial Poisson's ratios in MATORT are not supplied, the default is set to zero. Therefore, the behavior of the material during compaction is uncoupled. This means that straining in the (local) x-direction produces stresses only in the (local) x-direction, and not in the (local) y- or z-direction. The tables define the variation of the stress in a particular direction with the relative volume or the crush. The relative volume is defined as (current volume)/(initial volume) and varies from 1.0 (uncompressed) to 0.0 (zero volume). Crush is defined as one minus the relative volume and varies from 0.0 to 1.0. Since the tables should be defined with increasing x-values, it is convenient to use the default value for type, which is CRUSH. When defining the curves, care should be taken that the extrapolated values do not lead to negative yield stresses.
16. The elastic moduli (and the initial Poisson's ratios only if they are supplied) in MATORT vary linearly with the relative volume from their initial uncompacted values to the fully compacted ones.
17. When the material is fully compacted, its behavior becomes isotropic with an elastic perfectly plastic material characteristic.
18. If the TIDSR option is used, you can supply a table including strain-rate effects. Strain rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor; i.e.,

$$\dot{\varepsilon} = \sqrt{\varepsilon_{ij}^{dev} \cdot \varepsilon_{ij}^{dev}}$$

The y values in this table are factors with which the stresses in the other tables are multiplied to incorporate strain-rate effects.

19. The pressure-volume characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain and is defined as  $(1 - \frac{V}{V_0})$ , with  $\frac{V}{V_0}$  as the relative volume; or in terms of the volumetric (true) strain which is defined as:

$$\int_{V_0}^V \frac{dV}{V}$$

or

$$\ln\left(\frac{V}{V_0}\right)$$

The crush factor must be between 0 and 1. The volumetric strain must always be negative.

20. If the field for the value of CUTOFF is left blank, then this value is calculated from the yield function defined by the constants A0, A1, and A2. In case of a Mohr-Coulomb yield model, the cut-off pressure is calculated as the root of the pressure-yield stress curve. If the YIELD=1.0 is used, the cut-off pressure is calculated as the intersection point of the yield surface with the hydrostat (if only A0 is nonzero, then the cut-off pressure is set to -100K, where K is the bulk modulus). The cut-off pressure must be negative.
21. Either a minimum pressure (CUTOFF=3.0) or a failure pressure (CUTOFF=1.0 or 2.0) can be specified. In the first case, this corresponds to a tensile cutoff, where the pressure cannot fall below the minimum value. In the second case, if the pressure falls below the failure pressure the element fails and cannot carry tensile loading for the remainder of the analysis. Thus, the pressure can never become negative again. If CUTOFF=1.0 is used, the elements can physically fail, which means that the stresses are set to zero, but also the failure flag is used as in normal MATF models. If CUTOFF=2.0 is used, only the stresses are set to zero.
22. If YIELD=1.0 is used, the yield stress is determined by a Mohr-Coulomb model:

$$\sigma_y = \text{MIN}(A_0 + A_1 p, A_2)$$

If YIELD=2.0 is used, the yield surface in three-dimensional space is defined by  $s_s=0$  here

$$\Phi_s = \frac{1}{2} S_{ij} S_{ij} - (B_0 + B_1 p + B_2 p^2) = J_2 - (B_0 + B_1 p + B_2 p^2)$$

where  $S_{ij}$  are the deviatoric stresses and  $J_2$  is the second invariant of the stress deviation. The coefficients  $B_0$ ,  $B_1$  and  $B_2$  can be related to the coefficients  $A_0$ ,  $A_1$  and  $A_2$ . The relation between the coefficients depends on the YSTYP field as shown below.

If YSTYP=2.0, then

$$B_0 = A_0$$

$$B_1 = A_1$$

$$B_2 = A_2$$

Thus, the yield stress is defined as

$$\sigma_y = \sqrt{3(A_0 + A_1 p + A_2 p^2)}$$

If YSTYP=1.0, then

$$B_0 = \frac{1}{3}A_0^2$$

$$B_1 = \frac{2}{3}A_0 A_1$$

$$B_2 = \frac{1}{3}A_1^2$$

and A2 is ignored.

Thus, the yield stress is defined as

$$\sigma_y = A_0 + A_1 p$$

**MAT1F****Isotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT1 entry fields via TABLEDi entries.

**Format:**

| 1     | 2   | 3 | 4 | 5  | 6   | 7 | 8 | 9  | 10 |
|-------|-----|---|---|----|-----|---|---|----|----|
| MAT1F | MID | E | G | NU | RHO |   |   | GE |    |

**Example:**

|       |    |    |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|
| MAT1F | 33 | 15 |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)   |
| E          | Identification number of a TABLEDi; i=1 to 4 entry for the Young's modulus. (Integer > 0 or blank)  |
| G          | Identification number of a TABLEDi; i=1 to 4 entry for the shear modulus. (Integer > 0 or blank)  |
| NU         | Identification number of a TABLEDi; i=1 to 4 entry for the Poisson's ratio. (Integer > 0 or blank)  |
| GE         | Identification number of a TABLEDi; i=1 to 4 entry for the structural damping coefficient. (Integer > 0 or blank)   |
| RHO        | Identification number of a TABLEDi; i=1 to 4 entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only. |

- |  |  |
| --- | --- |
| MID | Material property identification number that matches the identification number on MAT1 entry. (Integer > 0) |
- |  |  |
| --- | --- |
| E | Identification number of a TABLEDi; i=1 to 4 entry for the Young's modulus. (Integer > 0 or blank) |
- |  |  |
| --- | --- |
| G | Identification number of a TABLEDi; i=1 to 4 entry for the shear modulus. (Integer > 0 or blank) |
- |  |  |
| --- | --- |
| NU | Identification number of a TABLEDi; i=1 to 4 entry for the Poisson's ratio. (Integer > 0 or blank) |
- |  |  |
| --- | --- |
| GE | Identification number of a TABLEDi; i=1 to 4 entry for the structural damping coefficient. (Integer > 0 or blank) |
- |  |  |
| --- | --- |
| RHO | Identification number of a TABLEDi; i=1 to 4 entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only. |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT1 entry.
- The user is reminded that on the MAT1 entry, if any one of the entries E, G, or, NU is left blank, they are connected by the relationship  $G=E/(2(1+NU))$ . Therefore, the user is cautioned that leaving one of these entries blank or only making one of these entries spatially, frequency, or temperature dependent may lead to unexpected results.
- Table references must be present for each item that is frequency dependent.
- Where applicable, secondary material entries such as the MATVE are supported through their MID field being associated with a primary material entry such as a MAT1 entry.

## MATF

## Material Failure Model

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis. Up to three criteria may be specified for each material. For progressive failure (ITYPE=2) only the first criterion will contribute to failure. The other two, if specified, will only provide output of failure index. SOL 400 (only with extended property entries), and all linear solution sequences between SOL101 and SOL112 (only with CPYRAM or CHEXA/PCOMPLS) are supported.

Format:

| 1    | 2                           | 3                           | 4                           | 5                           | 6                            | 7                            | 8                            | 9                           | 10  |
|------|-----------------------------|-----------------------------|-----------------------------|-----------------------------|------------------------------|------------------------------|------------------------------|-----------------------------|-----|
| MATF | MID                         | ITYPE                       |                             |                             |                              |                              |                              |                             |     |
|      | “CRI”                       | Criterion                   | V <sub>1</sub> <sup>1</sup> | V <sub>2</sub> <sup>1</sup> | V <sub>3</sub> <sup>1</sup>  | V <sub>4</sub> <sup>1</sup>  | V <sub>5</sub> <sup>1</sup>  | V <sub>6</sub> <sup>1</sup> | 1st |
|      | V <sub>7</sub> <sup>1</sup> | V <sub>8</sub> <sup>1</sup> | V <sub>9</sub> <sup>1</sup> | Find <sup>1</sup>           | V <sub>10</sub> <sup>1</sup> | V <sub>11</sub> <sup>1</sup> | V <sub>12</sub> <sup>1</sup> | W <sub>1</sub> <sup>1</sup> |     |
|      | W <sub>2</sub> <sup>1</sup> | W <sub>3</sub> <sup>1</sup> | W <sub>4</sub> <sup>1</sup> | W <sub>5</sub> <sup>1</sup> | W <sub>6</sub> <sup>1</sup>  | W <sub>7</sub> <sup>1</sup>  | W <sub>8</sub> <sup>1</sup>  | W <sub>9</sub> <sup>1</sup> |     |
|      | “PF”                        | A1                          | A2                          | A3                          | A4                           | A5                           |                              |                             |     |
|      | “CRI”                       | Criterion                   | V <sub>1</sub> <sup>2</sup> | V <sub>2</sub> <sup>2</sup> | V <sub>3</sub> <sup>2</sup>  | V <sub>4</sub> <sup>2</sup>  | V <sub>5</sub> <sup>2</sup>  | V <sub>6</sub> <sup>2</sup> | 2nd |
|      | V <sub>7</sub> <sup>2</sup> | V <sub>8</sub> <sup>2</sup> | V <sub>9</sub> <sup>2</sup> | Find <sup>2</sup>           | V <sub>10</sub> <sup>2</sup> | V <sub>11</sub> <sup>2</sup> | V <sub>12</sub> <sup>2</sup> | W <sub>1</sub> <sup>2</sup> |     |
|      | W <sub>2</sub> <sup>2</sup> | W <sub>3</sub> <sup>2</sup> | W <sub>4</sub> <sup>2</sup> | W <sub>5</sub> <sup>2</sup> | W <sub>6</sub> <sup>2</sup>  | W <sub>7</sub> <sup>2</sup>  | W <sub>8</sub> <sup>2</sup>  | W <sub>9</sub> <sup>2</sup> |     |
|      | “CRI”                       | Criterion                   | V <sub>1</sub> <sup>3</sup> | V <sub>2</sub> <sup>3</sup> | V <sub>3</sub> <sup>3</sup>  | V <sub>4</sub> <sup>3</sup>  | V <sub>5</sub> <sup>3</sup>  | V <sub>6</sub> <sup>3</sup> | 3rd |
|      | V <sub>7</sub> <sup>3</sup> | V <sub>8</sub> <sup>3</sup> | V <sub>9</sub> <sup>3</sup> | Find <sup>3</sup>           | V <sub>10</sub> <sup>3</sup> | V <sub>11</sub> <sup>3</sup> | V <sub>12</sub> <sup>3</sup> | W <sub>1</sub> <sup>3</sup> |     |
|      | W <sub>2</sub> <sup>3</sup> | W <sub>3</sub> <sup>3</sup> | W <sub>4</sub> <sup>3</sup> | W <sub>5</sub> <sup>3</sup> | W <sub>6</sub> <sup>3</sup>  | W <sub>7</sub> <sup>3</sup>  | W <sub>8</sub> <sup>3</sup>  | W <sub>9</sub> <sup>3</sup> |     |

Example 1 (3 Criteria, no progressive failure):

|      |       |       |       |       |       |       |       |       |     |
|------|-------|-------|-------|-------|-------|-------|-------|-------|-----|
| MATF | 100   | 0     |       |       |       |       |       |       |     |
|      | CRI   | 1     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. | 1st |
|      | 4500. | 4500. | 4500. |       |       |       |       |       |     |
| +    |       |       |       |       |       |       |       |       |     |
|      | CRI   | 7     | 2500. | 3000. | 1500. | 2000. |       |       | 2nd |
| +    | 1000. | 1000. |       |       |       |       |       |       |     |
|      | .11   | .06   | .1    | .05   | .075  | .03   | .03   | .03   |     |
|      | CRI   | 4     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. | 3rd |
|      | 4500. | 4500. | 4500. |       |       |       |       |       |     |

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criteria has been omitted.)

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default) |
| ITYPE      | Flag to invoke progressive failure. (Integer; Default = 0)   |

| Descriptor        | Meaning   |
|-------------------|---|
| 0                 | No progressive failure, compute failure indices only.   |
| 2                 | Gradual selective stiffness degradation.  |
| 3                 | Immediate selective stiffness degradation.  |
| “CRI”             | Enter the character string “CRI” to start input data for a failure criterion.   |
| CRI67             | Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows:<br>1000000*ITYPE+10000*C3+100*C2+C1<br>where C1, C2, C3 are integer values for the various Criteria listed.   |
| Criterion         | Select an integer corresponding to the failure criterion to be applied. (Integer; no Default)<br>Up to three failure criteria may be specified for each MID. <ul style="list-style-type: none"> <li>1 For maximum stress criterion. (See Remark 3.)</li> <li>2 For maximum strain criterion. (See Remark 4.)</li> <li>3 For Hill failure criterion. (See Remark 5.)</li> <li>4 For Hoffman failure criterion. (See Remark 6.)</li> <li>5 Tsai-Wu failure criterion. (See Remark 7.)</li> <li>7 Hashin failure criterion. (See Remark 8.)</li> <li>8 Puck failure criterion. (See Remark 9.)</li> <li>10 Hashin-Tape (See Remark 10.)</li> <li>11 Hashin-Fabric (See Remark 11.)</li> <li>13 User-defined criterion with user subroutine UFAIL See Remark 13.</li> </ul> See the respective remark for the meaning of material data. |
| $V_i^j$           | Stress limit $i$ for criterion $j$ . See Remarks for meaning and default.   |
| $W_i^j$           | Strain limit $i$ for criterion $j$ . See Remarks for meaning and default.   |
| Find <sup>j</sup> | Failure index scale factor for criterion $j$ . Used for criteria 3, 4, and 5. (Real > 0.0, or blank, Default = 1.0)   |
| “PF”              | Enter the character string “PF” to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line may be omitted. Only the first criterion uses the PF line. (Character; no Default)   |
| A1                | Residual stiffness fraction. For ITYPE=2, the stiffness is not reduced beyond this fraction of the initial stiffness. For ITYPE=3, this is the fraction of initial stiffness upon failure. (Real > 0.0, Default = 0.01)   |
| A2                | Leave blank except for Criterion values listed below.   |

| Descriptor | Meaning  |
|------------|--|
|            | Criterion=7 (Hashin) A2 is the factor for E2 reduction due to matrix compression failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix compression failure.   |
|            | Criterion=8 (Puck) - Same as for Criterion=7   |
|            | Criterion=10 (Hashin Tape) - Same as for Criterion=7   |
| A3         | Leave blank except for the Criterion values listed below.<br><br>Criterion=7 (Hashin) A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction.<br><br>Criterion=8 (Puck) - Same as for Criterion=7<br><br>Criterion=10 (Hashin Tape) - Same as for Criterion=7   |
| A4         | Leave blank except for the Criterion values listed below<br><br>Criterion=7 (Hashin) A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.<br><br>Criterion=8 (Puck) - Same as for Criterion=7<br><br>Criterion=10 (Hashin Tape) - Factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure. |
| A5         | Leave blank except for the Criterion values listed below<br><br>Criterion=7 (Hashin) A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0. It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.<br><br>Criterion=8 (Puck) - Same as for Criterion=7<br><br>Criterion=10 (Hashin Tape) - Same as for Criterion=7   |

**Remarks:**

1. The MATF Bulk Data entry contains supplementary data for failure index calculation and failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate a progressive failure process.

2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied; that is, the calculated failure index exceeds 1.0. Upon failure, the elastic moduli are reduced. This is done differently depending on failure criterion. The behavior up to the failure point is linear elastic. After failure the behavior is still elastic but with a different stiffness. The option should not be combined with other material nonlinearities like plasticity.
3. According to the Maximum Stress Criterion, six failure indices  $F_i$  are calculated as follows:

$$F_1 = \begin{cases} \frac{\sigma_x}{V_1} & \text{for } \sigma_x > 0 \\ \frac{-\sigma_x}{V_2} & \text{for } \sigma_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\sigma_y}{V_3} & \text{for } \sigma_y > 0 \\ \frac{-\sigma_y}{V_4} & \text{for } \sigma_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\sigma_z}{V_5} & \text{for } \sigma_z > 0 \\ \frac{-\sigma_z}{V_6} & \text{for } \sigma_z < 0 \end{cases}$$

$$F_4 = \frac{\tau_{xy}}{V_7}$$

$$F_5 = \frac{\tau_{yz}}{V_8}$$

$$F_6 = \frac{\tau_{zx}}{V_9}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

$V_2$  : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default  $V_1$

$V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

$V_4$  : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default  $V_3$

$V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

$V_6$  : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default  $V_5$

$V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default

$V_8$  : maximum allowable  $yz$  shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable  $zx$  shear stress. Real > 0.0 or blank; Default  $V_7$

- According to the Maximum Strain Criterion, 6 failure indices  $F_i$  are calculated as follows:

$$F_1 = \begin{cases} \frac{\varepsilon_x}{W_1} & \text{for } \varepsilon_x > 0 \\ \frac{-\varepsilon_x}{W_2} & \text{for } \varepsilon_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\varepsilon_y}{W_3} & \text{for } \varepsilon_y > 0 \\ \frac{-\varepsilon_y}{W_4} & \text{for } \varepsilon_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\varepsilon_z}{W_5} & \text{for } \varepsilon_z > 0 \\ \frac{-\varepsilon_z}{W_6} & \text{for } \varepsilon_z < 0 \end{cases}$$

$$F_4 = \frac{\varepsilon_{xy}}{W_7}$$

$$F_5 = \frac{\varepsilon_{yz}}{W_8}$$

$$F_6 = \frac{\varepsilon_{zx}}{W_9}$$

where the indices (x, y, z) denote material coordinate direction.

$W_1$  : maximum allowable strain in the x-direction in tension. Real > 0.0, no Default.

$W_2$  : maximum allowable strain in the x-direction in compression. Real > 0.0 or blank; Default  $W_1$

$W_3$  : maximum allowable strain in the y-direction in tension. Real > 0.0, no Default.

$W_4$  : maximum allowable strain in the y-direction in compression. Real > 0.0 or blank; Default  $W_3$

$W_5$  : maximum allowable strain in the z-direction in tension. Real > 0.0; no Default

$W_6$  : maximum allowable strain in the z-direction in compression. Real > 0.0 or blank; Default  $W_5$

$W_7$  : maximum allowable shear strain in the xy plane. Real > 0.0 no Default

$W_8$  : maximum allowable shear strain in the yz plane. Real > 0.0 or blank; Default  $W_7$

$W_9$  : maximum allowable shear strain in the zx plane. Real > 0.0 or blank; Default  $W_7$

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. A single failure index is calculated as:

$$F = \left[ \frac{\sigma_x^2}{V_1^2} + \frac{\sigma_y^2}{V_3^2} + \frac{\sigma_z^2}{V_5^2} - \left( \frac{1}{V_1^2} + \frac{1}{V_3^2} - \frac{1}{V_5^2} \right) \sigma_x \sigma_y - \left( \frac{1}{V_1^2} + \frac{1}{V_5^2} - \frac{1}{V_3^2} \right) \sigma_x \sigma_z \right. \\ \left. - \left( \frac{1}{V_3^2} + \frac{1}{V_5^2} - \frac{1}{V_1^2} \right) \sigma_y \sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} \right] / \text{Find}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

$V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

$V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

$V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default

$V_8$  : maximum allowable yz shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable zx shear stress. Real > 0.0 or blank; Default  $V_7$

6. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion. A single failure index F is calculated as:

$$F = (C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left(\frac{1}{V_1} - \frac{1}{V_2}\right)\sigma_x + \left(\frac{1}{V_3} - \frac{1}{V_4}\right)\sigma_y + \left(\frac{1}{V_5} - \frac{1}{V_6}\right)\sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2}) / \text{Find}$$

with

$$C_x = \frac{1}{2} \left( \frac{1}{V_1 V_2} + \frac{1}{V_3 V_4} - \frac{1}{V_5 V_6} \right)$$

$$C_y = \frac{1}{2} \left( \frac{1}{V_3 V_4} + \frac{1}{V_5 V_6} - \frac{1}{V_1 V_2} \right)$$

$$C_z = \frac{1}{2} \left( \frac{1}{V_5 V_6} + \frac{1}{V_1 V_2} - \frac{1}{V_3 V_4} \right)$$

where the indices (x, y, z) denote material coordinate direction.

- V<sub>1</sub>: maximum allowable stress in the x-direction in tension (Real > 0.0, no Default).
- V<sub>2</sub>: maximum allowable stress in the x-direction in compression (Real > 0.0 or blank; Default V<sub>1</sub>)
- V<sub>3</sub>: maximum allowable stress in the y-direction in tension (Real > 0.0, no Default).
- V<sub>4</sub>: maximum allowable stress in the y-direction in compression (Real > 0.0 or blank; Default V<sub>3</sub>)
- V<sub>5</sub>: maximum allowable stress in the z-direction in tension (Real > 0.0, no Default).
- V<sub>6</sub>: maximum allowable stress in the z-direction in compression (Real > 0.0 or blank; Default V<sub>5</sub>)
- V<sub>7</sub>: maximum allowable in-plane shear stress (Real > 0.0, no Default).
- V<sub>8</sub>: maximum allowable yz shear stress (Real > 0.0, no Default V<sub>7</sub>).
- V<sub>9</sub>: maximum allowable zx shear stress (Real > 0.0, no Default V<sub>7</sub>).

7. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion. A single failure index F is calculated as:

$$F = \left[ \left( \frac{1}{V_1} - \frac{1}{V_2} \right) \sigma_x + \left( \frac{1}{V_3} - \frac{1}{V_4} \right) \sigma_y + \left( \frac{1}{V_5} - \frac{1}{V_6} \right) \sigma_z + \frac{\sigma_x^2}{V_1 V_2} + \frac{\sigma_y^2}{V_3 V_4} + \frac{\sigma_z^2}{V_5 V_6} + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} + 2V_{10}\sigma_x\sigma_y + 2V_{11}\sigma_y\sigma_z + 2V_{12}\sigma_x\sigma_z \right] / \text{Find}^j$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

$V_2$  : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default  $V_1$

$V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

$V_4$  : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default  $V_3$

$V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

$V_6$  : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default  $V_5$

$V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default

$V_8$  : maximum allowable yz shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable zx shear stress. Real > 0.0 or blank; Default  $V_7$

$V_{10}$  : interactive strength constant for the xy plane. Real; no Default

$V_{11}$  : interactive strength constant for the yz plane. Real; no Default

$V_{12}$  : interactive strength constant for the zx plane. Real; no Default

The following restrictions apply to  $V_{10}$ ,  $V_{11}$  and  $V_{12}$ :

$$V_{10}^2 < \frac{1}{V_1 V_2 V_3 V_4} \quad V_{11}^2 < \frac{1}{V_3 V_4 V_5 V_6} \quad V_{12}^2 < \frac{1}{V_1 V_2 V_5 V_6}$$

8. For the Hashin criterion four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \frac{1}{V_{10}^2}(\tau_{xy}^2 + \tau_{xz}^2) & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{|\sigma_x|}{V_2} & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 + \frac{1}{V_{11}^2}(\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2}(\sigma_{xy}^2 + \sigma_{xz}^2) & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \leq 0 \end{cases}$$

$$\begin{aligned}
 & \left( \frac{1}{V_4} \left( \left( \frac{V_4}{2V_{11}} \right)^2 - 1 \right) (\sigma_y + \sigma_z) + \frac{1}{4V_{11}^2} (\sigma_y + \sigma_z)^2 + \right. \\
 F_4 = & \left. \frac{1}{V_{11}^2} (\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2} (\sigma_{xy}^2 + \sigma_{xz}^2) \right) & (\sigma_y + \sigma_z > 0) \\
 & 0 & \sigma_y + \sigma_z \geq 0
 \end{aligned}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum fiber tensile stress. Real > 0.0, no Default.

$V_2$  : maximum fiber compressive stress. Real > 0.0 or blank; Default  $V_1$

$V_3$  : maximum matrix tensile stress. Real > 0.0, no Default.

$V_4$  : maximum matrix compressive stress. Real > 0.0 or blank; Default  $V_3$ .

$V_{10}$  : maximum in plane shear stress. Real > 0.0 no Default

$V_{11}$  : maximum transverse shear stress; Default  $V_{10}$

9. For the Puck failure criterion five failure indices are calculated for the plane stress case and four for the general 3-D case.

The first two failure indices are related to fiber failure and apply to both plane stress and 3-D case:

Tensile fiber mode, for  $\sigma_x > 0$

$$F_1 = \frac{\sigma_x}{V_1}$$

Compressive fiber mode, for  $\sigma_x < 0$

$$F_2 = \frac{|\sigma_x|}{V_1}$$

For plane stress case, the next three failure indices are related to the matrix failure and are given as follows:

$$F_3 = \sqrt{\left( \frac{\sigma_{xy}}{V_{10}} \right)^2 + \left( 1 - W_2 \frac{V_3}{V_{10}} \right)^2 \left( \frac{\sigma_y}{V_3} \right)^2} + W_2 \frac{\sigma_y}{V_{10}} \quad \text{for } \sigma_y > 0$$

$$F_4 = \frac{1}{V_{10}} \left( \sqrt{\sigma_{xy}^2 + (W_1 \sigma_y)^2} + W_1 \sigma_y \right) \quad \text{for } \sigma_y < 0 \text{ and } \left( 0 \leq \left| \frac{\sigma_y}{\sigma_{xy}} \right| \leq \frac{R_1}{R_2} \right)$$

$$F_5 = \left( \frac{\sigma_{xy}}{2(1 + W_3 V_{10})} \right)^2 + \left( \frac{\sigma_y}{V_4} \right)^2 \frac{V_4}{|\sigma_y|} \quad \text{for } \sigma_y < 0 \text{ and } \left( 0 \leq \left| \frac{\sigma_{xy}}{\sigma_y} \right| \leq \frac{R_2}{R_1} \right)$$

where the parameters,  $R_1$  and  $R_2$  are as follows:

$$R_1 = \frac{V_{10}}{2(1 + W_1)} \left( \sqrt{1 + 2W_1 \frac{V_4}{V_{10}}} - 1 \right)$$

$$R_2 = V_{10} \sqrt{1 + 2W_3}$$

Failure index for 3-D case is calculated based on stress,  $\sigma_n(\theta)$ ,  $\sigma_{nt}(\theta)$  and  $\sigma_{n1}(\theta)$  on an arbitrary plane with inclination angle  $\theta$ . The stresses are given by:

$$\sigma_n = \sigma_y \cos^2 \theta + \sigma_z \sin^2 \theta + 2\sigma_{yz} \sin \theta \cos \theta$$

$$\sigma_{nt} = (\sigma_z - \sigma_y) \sin \theta \cos \theta + \sigma_{yz} (\cos^2 \theta - \sin^2 \theta)$$

$$\sigma_{n1} = \sigma_{zx} \sin \theta + \sigma_{xy} \cos \theta$$

For 3-D case, the next two failure indices related to matrix failure are given by:

$$F_3 = \sqrt{\left(\frac{1}{V_3} - P_1\right)^2 \sigma_n^2 + \left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2} + P_1 \sigma_n \quad \text{for } \sigma_n \geq 0$$

$$F_4 = \sqrt{\left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2 + (P_2 \sigma_n)^2} + P_2 \sigma_n \quad \text{for } \sigma_n \geq 0$$

where  $P_1$  and  $P_2$  are given as:

$$P_1 = \left(\frac{W_4}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_2}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

$$P_2 = \left(\frac{W_3}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_1}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

The following material parameters are used:

$V_1$  = Maximum fiber tensile stress (Real > 0.0; no Default)

$V_2$  = Maximum fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )

$V_3$  = Maximum matrix tensile stress (Real > 0.0; no Default)

$V_4$  = Maximum matrix compressive stress (Real > 0.0 or blank; Default  $V_3$ )

$V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)

$W_1 = p12c$ , slope 1 of failure envelope (Real > 0.0; no Default)

$W_2 = p12t$ , slope 2 of failure envelope (Real > 0.0; Default  $W_1$ )

$W_3 = p23c$ , slope 3 of failure envelope (Real > 0.0 or blank; Default; calculated internally; this is recommended for plane stress)

$W_4 = p23t$ , slope 4 of failure envelope (Real > 0.0; Default  $W_3$ , not used for plane stress)

- The Hashin-Tape criterion is a variant of the Hashin criterion adapted for tape type of materials. Four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\left(\frac{V_4}{2V_{11}}\right)^2 - 1\right) \frac{(\sigma_y + \sigma_z)}{V_4} + \frac{1}{4V_{11}^2}(\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left(\left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 + V_5 \left(\frac{\sigma_x}{V_6}\right)^2\right) & (\sigma_y + \sigma_z < 0) \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

$V_1$  = Maximum tape fiber tensile stress (Real > 0.0; no Default)

$V_2$  = Maximum tape fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )

$V_3$  = Maximum tape cross-fiber tensile stress (Real > 0.0; no Default)

$V_4$  = Maximum tape cross-fiber compressive stress (Real > 0.0 or blank; Default  $V_3$ )

$V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)

$V_{11}$  = Maximum transverse shear stress. (Real > 0.0 or blank; Default  $V_{10}$ )

$V_{12}$  = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default  $V_{11}$ )

$V_5$  = Maximum fiber tensile stress for matrix compression (Real > 0.0 or blank; required if

$V_6 = 1.0$ , otherwise not used)

$V_6$  = Contribution factor for  $V_5$  (Real 0.0 or 1.0; Default = 0.0)

11. The Hashin-Fabric criterion is a variant of the Hashin criterion adapted for fabric type of materials. Six failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \left(\frac{\sigma_y}{V_3}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\frac{\sigma_y}{V_4}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{12}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_5 = \begin{cases} \left(\frac{\sigma_z}{V_5}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

$$F_6 = \begin{cases} \left(\frac{\sigma_z}{V_6}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

$V_1$  = Maximum first fiber tensile stress (Real > 0.0; no Default)

$V_2$  = Maximum first fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )

$V_3$  = Maximum second cross-fiber tensile stress (Real > 0.0; no Default)

$V_4$  = Maximum second cross-fiber compressive stress (Real > 0.0 or blank; Default  $V_3$ )

$V_5$  = Maximum thickness tensile stress (Real > 0.0; no Default)

$V_6$  = Maximum thickness compressive stress ((Real > 0.0 or blank; Default =  $V_5$ )

$V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)

$V_{11}$  = Maximum transverse shear stress. (Real > 0.0 or blank; Default  $V_{10}$ )

$V_{12}$  = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default  $V_{11}$ )

12. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.

13. User subroutine UFAIL is used along with option MATUDS.
14. In SOL 400, MATF is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATF data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.  
For non-composite material with MATF, only the first failure criteria is calculated and its failure index is outputted in NLOUT.
15. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.
16. Stress limits such as ST, SC, SS, X<sub>t</sub>, X<sub>c</sub>, Y<sub>t</sub>, Y<sub>c</sub> in the MAT1, MAT2 and MAT8 entries are not used in SOL 400.
17. Stress and/or strain allowables in all directions should be defined if the default is not appropriate. Be sure not to set some of these values to low numbers.
18. If MAT8 with Hashin is used in SOL 400 run with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

**MATF****Material Failure Model - SOL 600**

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600. Up to three criterion may be specified for each material if failure indices are desired (ITYPE=0). For progressive failure (ITYPE=2) only one criterion should normally be specified. Even if failure indices (no progressive failure) is the objective of the analysis, it is recommended that only one criterion per material be specified to simplify post processing.

**Format:**

| 1              | 2              | 3              | 4              | 5               | 6               | 7               | 8              | 9   | 10 |
|----------------|----------------|----------------|----------------|-----------------|-----------------|-----------------|----------------|-----|----|
| MATF           | MID            | ITYPE          | SB             | UFAIL           |                 |                 |                |     |    |
| "CRI"          | Criteria       | V <sub>1</sub> | V <sub>2</sub> | V <sub>3</sub>  | V <sub>4</sub>  | V <sub>5</sub>  | V <sub>6</sub> | 1st |    |
| V <sub>7</sub> | V <sub>8</sub> | V <sub>9</sub> | Find           | V <sub>10</sub> | V <sub>11</sub> | V <sub>12</sub> | Ext            |     |    |
| Exc            | Eyt            | Eyc            | Ezt            | Ezc             | Gxy             | Gyz             | Gzx            |     |    |
| "PF"           | A1             | A2             | A3             | A4              | A5              | IC1             | IC2            |     |    |
| IC3            | IC4            | IC5            | IC6            | CI7             | IC8             | IC9             |                |     |    |
| "CRI"          | Criteria       | Xt             | Xc             | Yt              | Yc              | Zt              | Zc             | 2nd |    |
| Sxy            | Syz            | Szx            | Find           | Fxy             | Fyz             | Fzx             | Ext            |     |    |
| Exc            | Eyt            | Eyc            | Ezt            | Ezc             | Gxy             | Gyz             | Gzx            |     |    |
| "CRI"          | Criteria       | Xt             | Xc             | Yt              | Yc              | Zt              | Zc             | 3rd |    |
| Sxy            | Syz            | Szx            | Find           | Fxy             | Fyz             | Fzx             | Ext            |     |    |
| Exc            | Eyt            | Eyc            | Ezt            | Ezc             | Gxy             | Gyz             | Gzx            |     |    |

**Example 1 (3 Criteria, no progressive failure):**

|      |       |       |       |       |       |       |       |       |     |
|------|-------|-------|-------|-------|-------|-------|-------|-------|-----|
| MATF | 100   | 0     |       |       |       |       |       |       |     |
|      | CRI   | 1     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. | 1st |
|      | 4500. | 4500. | 4500. |       |       |       |       |       |     |
| +    |       |       |       |       |       |       |       |       |     |
|      | CRI   | 2     |       |       |       |       |       |       | 2nd |
| +    |       |       |       |       |       |       |       |       |     |
|      | .11   | .06   | .1    | .05   | .075  | .03   | .03   | .03   |     |
|      | CRI   | 4     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. | 3rd |
|      | 4500. | 4500. | 4500. | 0.90  |       |       |       |       |     |

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)

### Example 2 (with progressive failure):

|      |       |       |       |       |       |       |       |       |     |
|------|-------|-------|-------|-------|-------|-------|-------|-------|-----|
| MATF | 100   | 2     |       |       |       |       |       |       |     |
|      | CRI   | 1     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. | 1st |
|      | 4500. | 4500. | 4500. |       |       |       |       |       |     |
| +    |       |       |       |       |       |       |       |       |     |
|      | PF    | .001  |       |       |       |       | 1     | 1     |     |
| +    | 1     | 1     |       |       |       |       |       |       |     |

### Alternate Format:

| 1    | 2              | 3              | 4              | 5              | 6               | 7               | 8               | 9              | 10 |
|------|----------------|----------------|----------------|----------------|-----------------|-----------------|-----------------|----------------|----|
| MATF | MID            | CRI67          | V <sub>1</sub> | V <sub>2</sub> | V <sub>3</sub>  | V <sub>4</sub>  | V <sub>5</sub>  | V <sub>6</sub> |    |
|      | V <sub>7</sub> | V <sub>8</sub> | V <sub>8</sub> | Find           | V <sub>10</sub> | V <sub>11</sub> | V <sub>12</sub> | Ext            |    |
|      | Exc            | Eyt            | Eyc            | Ezt            | Ezc             | Gxy             | Gyz             | Gzx            |    |

### Alternate Example Format:

|      |       |       |       |       |       |       |       |       |  |
|------|-------|-------|-------|-------|-------|-------|-------|-------|--|
| MATF | 100   | 1     | 2500. | 4000. | 2500. | 4000. | 2000. | 3000. |  |
|      | 4500. | 4500. | 4500. |       |       |       |       |       |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default)   |
| ITYPE      | Flag to invoke progressive failure. (Integer; Default = 0)   |
| 0          | No progressive failure compute failure indices only.<br>(Default)  |
| 1          | Standard (original) Marc method.   |
| 2          | Gradual selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)  |
| 3          | Immediate selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)  |
| SB         | Allowable shear stress of bonding material between layers (composites only). To use SB in the stiffness formulation of composites, include PARAM,MINCLDSB,1 in the bulk data. (Real, no Default)   |
| UFAIL      | Enter the string "UFAIL" if the ufail.f user subroutine is to be used to define your own failure conditions. If UFAIL is entered, leave fields 3 and 4 blank and do not enter any continuation lines. UFAIL is not available using the alternate format. (Character; Default is blank,). The file ufail.f must be in the same directory as the Nastran input file and must be in lower case. |
| "CRI"      | For the primary failure model, warning may lead to slow convergence.   |

| Descriptor        | Meaning   |
|-------------------|---|
| CRI67             | Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows:<br>$1000000*\text{ITYPE}+10000*\text{C3}+100*\text{C2}+\text{C1}$<br>where C1, C2, C3 are integer values for the various Criteria are listed.   |
| Criteria          | Select an integer corresponding to the failure criterion to be applied. (Integer; no Default)<br>Up to three failure criterion may be specified for each MID for SOL 600. <ul style="list-style-type: none"> <li>1 for maximum stress criterion. (See Remark 3.)</li> <li>2 for maximum strain criterion. (See Remark 4.)</li> <li>3 for Hill failure criterion. (See Remark 5.)</li> <li>4 for Hoffman failure criterion. (See Remark a.)</li> <li>5 Tsai-Wu failure criterion. (See Remark b.)</li> <li>7 Hashin failure criterion. (See Remark 8.)</li> <li>8 Puck failure criterion (See Remark 9.)</li> <li>9 User-defined failure criterion. (See Remark 8.)</li> <li>10 Hashin-Tape (See Remark 10.)</li> <li>11 Hashin-Fabric (See Remark 11.)</li> </ul> For Hashin models or Puck model, see Remarks 9., 8., 10., 11. for meaning of material data. |
| $V_1 = X_t$       | Maximum tensile stress in x-direction. (Real, 0.0, or blank, no Default)  |
| $V_2 = X_c$       | Maximum compressive stress (absolute value) in x-direction (Real, 0.0, or blank, Default = $X_t$ )  |
| $V_3 = Y_t$       | Maximum tensile stress in y-direction. (Real, 0.0, or blank, no Default)  |
| $V_4 = Y_c$       | Maximum compressive stress (absolute value) in y-direction. (Real, 0.0, or blank, Default = $Y_t$ )   |
| $V_5 = Z_t$       | Maximum tensile stress in z-direction. (Real, 0.0, or blank, no Default)  |
| $V_6 = Z_c$       | Maximum compressive stress (absolute value) in z-direction. (Real, 0.0, or blank, Default = $Z_t$ )   |
| $V_7 = S_{xy}$    | Maximum shear stress in xy-plane. (Real, 0.0, or blank, no Default)   |
| $V_8 = S_{yz}$    | Maximum shear stress in yz-plane. (Real, 0.0, or blank, Default = $S_{xy}$ for criterion 5)   |
| $V_9 = S_{zx}$    | Maximum shear stress in zx-plane (Real, 0.0, or blank, Default = $S_{xy}$ for criterion 5)  |
| Find              | Failure index. See Remarks 5.-a. (Real, 0.0, or blank, Default = 1.0)   |
| $V_{10} = F_{xy}$ | Interactive strength constant for xy-plane. (Real, 0.0, or blank, Default = $\frac{1}{2}\sqrt{\frac{1}{X_t X_c} \frac{1}{Y_t Y_c}}$ for criterion 5)  |

| Descriptor                | Meaning   |
|---------------------------|---|
| $V_{11} = F_{yz}$         | Interactive strength constant for yz-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2} \sqrt{\frac{1}{Y_t Y_c} \frac{1}{Z_t Z_c}}$ for criteria 5)   |
| $V_{12} = F_{zx}$         | Interactive strength constant for zx-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2} \sqrt{\frac{1}{Z_t Z_c} \frac{1}{X_t X_c}}$ for criterion 5).   |
| Ext                       | Maximum tensile strain in x-direction. (Real, 0.0, or blank, no Default)  |
| Exc                       | Maximum compressive strain (absolute value) in x-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)  |
| Eyt                       | Maximum tensile strain in y-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)   |
| Eyc                       | Maximum compressive strain (absolute value) in y-direction. (Real, 0.0, or blank, Default = Eyt for criterion 2)  |
| Ezt                       | Maximum tensile strain in z-direction. ( Real, 0.0, or blank, Default = Ext for criterion 2)  |
| Ezc                       | Maximum compressive strain (absolute value) in z-direction. (Real, 0.0, or blank, Default = Ezt for criterra 2)   |
| Gxy                       | Maximum shear strain in xy-plane. (Real, 0.0, or blank, no Default)   |
| Gyz                       | Maximum shear strain in yz-plane. (Real, 0.0, or blank, Default = Gxy for criterion 2)  |
| Gzx                       | Maximum shear strain in zx-plane. (Real, 0.0, or blank, Default =Gxy for criterion 2)   |
| "PF"                      | Enter the character string "PF" to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line and the line following the PF line may be omitted (Character, no Default)  |
| A1                        | Residual stiffness fraction. For Criteria=3, this is the fraction of initial stiffness upon failure. For Criteria=2, the stiffness is not reduced more than this fraction (Real, Default = 0.01)  |
| A2                        | Must be 0.0 or blank except for the Criteria values listed below  |
| Criteria=7 (Hashin)       | A2 is the factor for E2 reduction due to matrix comporession failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix comporession failure. |
| Criteria=10 (Hashin Tape) | Same as for Criteria=7  |
| Criteria=8 (Puck)         | Same as for Criteria=7  |

| Descriptor | Meaning   |
|------------|---|
| A3         | <p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p>  |
| A4         | <p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=10 (Hashin Tape) Factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p> |
| A5         | <p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin) A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0. It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p>   |
| IC1        | <p>For all Criteria values except those listed below, IC1 is set to 1 if failure in the positive x-direction is critical (leads to element deactivation).</p> <p>Criteria=7 (Hashin) Set IC1=1 if fiber tension is critical (leads to element deactivation).</p> <p>Criteria=10 (Hashin Tape) Same as for Criteria=7</p> <p>Criteria=8 (Puck) Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>  |

| Descriptor | Meaning  |
|------------|--|
| IC2        | <p>For all Criteria values except those listed below, IC2 is set to 1 if failure in the negative x-direction is critical</p> <p>Criteria=7 (Hashin)      Set IC2=1 if fiber compression is critical</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>                              |
| IC3        | <p>For all Criteria values except those listed below, IC3 is set to 1 if failure in the positive y-direction is critical</p> <p>Criteria=7 (Hashin)      Set IC3=1 if matrix tension is critical</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>                                 |
| IC4        | <p>For all Criteria values except those listed below, IC4 is set to 1 if failure in the negative y-direction is critical</p> <p>Criteria=7 (Hashin)      Set IC4=1 if matrix compression is critical</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Set IC4=1 if matrix compression mode B is critical</p> <p>(Integer, Default = 0)</p> |
| IC5        | <p>For all Criteria values except those listed below, IC5 is set to 1 if failure in the positive z-direction is critical</p> <p>Criteria=7 (Hashin)      Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)      Not used leave blank</p> <p>Criteria=8 (Puck)      Set IC5=1 if matrix compression mode C is critical</p> <p>(Integer, Default = 0)</p>                         |
| IC6        | <p>For all Criteria values except those listed below, IC6 is set to 1 if failure in the negative z--direction is critical</p> <p>Criteria=7 (Hashin)      Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)      Not used leave blank</p> <p>Criteria=8 (Puck)      Not used leave blank</p> <p>(Integer, Default = 0)</p>  |

| Descriptor | Meaning   |
|------------|---|
| IC7        | For all Criteria values except those listed below, IC7 is set to 1 if failure in the xy plane is critical<br><br>Criteria=7 (Hashin)              Not used, leave blank<br>Criteria=10 (Hashin Tape)      Not used leave blank<br>Criteria=8 (Puck)              Not used leave blank<br>(Integer, Default = 0) |
| IC8        | For all Criteria values except those listed below, IC8 is set to 1 if failure in the yz plane is critical<br><br>Criteria=7 (Hashin)              Not used, leave blank<br>Criteria=10 (Hashin Tape)      Not used leave blank<br>Criteria=8 (Puck)              Not used leave blank<br>(Integer, Default = 0) |
| IC9        | For all Criteria values except those listed below, IC9 is set to 1 if failure in the zx plane is critical<br><br>Criteria=7 (Hashin)              Not used, leave blank<br>Criteria=10 (Hashin Tape)      Not used leave blank<br>Criteria=8 (Puck)              Not used leave blank<br>(Integer, Default = 0) |

**Remarks:**

1. The MATF Bulk Data entry contains supplementary data for failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate progressive failure process.
2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied. Upon failure, the elastic modulus reduces to 10% of the original modulus if there is only one value of modulus as in isotropic material or in a beam or truss element. If it pertains to an orthotropic material, all of the material moduli at the integration point are reduced to the lowest modulus specified. The behavior up to the failure point is linear elastic even if an elasto-plastic material is specified, which is followed by a nonlinear behavior for the post-failure analysis. If the initial yield stress is less than the allowable maximum stress, the failure criterion will be ignored.
3. According to the Maximum Stress Criterion, the material fails when any of the stress components (9 components including 6 normal stress components in tension and compression, and three shear stress components) exceeds the maximum allowable stress:

$$|\sigma_{ij}| > X_t, X_c, Y_t, \dots, S_{xy}, \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.

4. According to the Maximum Strain Criterion, the material fails when any of the strain components (9 components including 6 normal strain components in tension and compression, and three shear strain components) exceeds the maximum allowable strain:

$$|\epsilon_{ij}| > \text{Ext, } \text{Exc, } \text{Ext, } \dots, \text{Gxy, } \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. The failure is determined based on

$$\frac{\sigma_x^2}{X^2} + \frac{\sigma_y^2}{Y^2} + \frac{\sigma_z^2}{Z^2} - \left( \frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2} \right) \sigma_x \sigma_y - \left( \frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2} \right) \sigma_y \sigma_z$$

$$- \left( \frac{1}{Z^2} + \frac{1}{X^2} - \frac{1}{Y^2} \right) \sigma_z \sigma_x + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{\text{ind}}$$

in which X, Y, Z, S<sub>xy</sub>, S<sub>yz</sub>, S<sub>zx</sub> are maximum allowable stresses and F<sub>ind</sub> is the failure index prescribed by the user.

- a. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion, i.e.,

$$C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left( \frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_x$$

$$+ \left( \frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_y + \left( \frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_z + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{\text{ind}}$$

with

$$C_x = \frac{1}{2} \left( \frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c} \right)$$

$$C_y = \frac{1}{2} \left( \frac{1}{Y_t Y_c} + \frac{1}{Z_t Z_c} - \frac{1}{X_t X_c} \right)$$

$$C_z = \frac{1}{2} \left( \frac{1}{Z_t Z_c} + \frac{1}{X_t X_c} - \frac{1}{Y_t Y_c} \right)$$

in which  $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$  are maximum allowable stresses and  $F_{ind}$  is the failure index, prescribed by the user.

- b. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion:

$$\left( \frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_x + \left( \frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_y + \left( \frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_z + \frac{\sigma_x^2}{X_t X_c} + \frac{\sigma_y^2}{Y_t Y_c} + \frac{\sigma_z^2}{Z_t Z_c}$$

$$+ \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} + 2F_{xy}\sigma_x\sigma_y + 2F_{yz}\sigma_y\sigma_z + 2F_{zx}\sigma_x\sigma_z > F_{ind}$$

in which  $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$  are maximum allowable stresses,  $F_{xy}, F_{yz}, F_{zx}$  are interactive strength constants, and  $F_{ind}$  is the failure index, prescribed by the user.

- 6. For the Hashin criterion the inputs are used:

$V_1$  = Maximum fiber tensile stress

$V_2$  = Maximum fiber compressive stress

$V_3$  = Maximum matrix tensile stress

$V_4$  = Maximum matrix compressive stress

$V_7$  = Maximum in-plane shear stress

$V_8$  = Maximum transverse shear stress

All other variables should be set to zero

- 7. For the Puck failure criterion the following inputs are used:

$V_1$  = Maximum fiber tensile stress

$V_2$  = Maximum fiber compressive stress

$V_3$  = Maximum matrix tensile stress

$V_4$  = Maximum matrix compressive stress

$V_7$  = Maximum in-plane shear stress

Ext = p12c, slope 1 of failure envelope  
Exc = p12t, slope 2 of failure envelope  
Eyt = p23c, slope 3 of failure envelope  
Eyc = p23t, slope 4 of failure envelope

8. For the user-defined failure criterion, leave all fields blank except fields 1-3 of the primary entry (continuation entries are not required).
9. For the Hashin-Tape criterion the following inputs are used:

$V_1$  = Maximum tape fiber tensile stress  
 $V_2$  = Maximum tape fiber compressive stress  
 $V_3$  = Maximum tape cross-fiber tensile stress  
 $V_4$  = Maximum tape cross-fiber compressive stress  
 $V_7$  = Maximum in-plane shear stress  
 $V_8$  = Maximum transverse shear stress  
 $V_9$  = Maximum z-x transverse shear stress  
 $V_5$  = Maximum fiber tensile stress for matrix compression  
 $V_6$  = Contribution factor for Zt (Real 0.0 or 1.0; Default = 0.0)

All other variables should be set to zero

For the Hashin-Fabric criterion the following inputs are used:

$V_1$  = Maximum first fiber tensile stress  
 $V_2$  = Maximum first fiber compressive stress  
 $V_3$  = Maximum second cross-fiber tensile stress  
 $V_4$  = Maximum second cross-fiber compressive stress  
 $V_5$  = Maximum thickness tensile stress  
 $V_6$  = Maximum thickness compressive stress  
 $V_7$  = Maximum in-plane shear stress  
 $V_8$  = Maximum transverse shear stress  
 $V_9$  = Maximum z-x transverse shear stress

All other variables should be set to zero

10. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.

#### Notes:

1. This Bulk Data entry accommodates Marc's input data under the model definition option FAIL DATA.
2. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.

3. For the alternate format the third line may be omitted if no data is needed.
4. For Nastran versions prior to MSC 2007, the alternate format was the only available option. These older versions used ITYPE=1 and a packed list for CRI67 with the format 100\*C3+10\*C2+C1. ITYPE=1 was an older progressive failure method that is no longer recommended. If these jobs are re-run using MSC 2007, the CRI67 field should be changed as described above or the new primary format used.
5. Stress limits such as ST, SC, SS, X<sub>t</sub>, X<sub>c</sub>, Y<sub>t</sub>, Y<sub>c</sub> in the MAT1, MAT2 and MAT8 entries are not used in SOL 600.

**MATF****Material Failure Model - SOL 700**

Specifies failure model properties. Use SOL700 only.

**Format:**

| 1    | 2     | 3                   | 4     | 5     | 6     | 7    | 8    | 9     | 10 |
|------|-------|---------------------|-------|-------|-------|------|------|-------|----|
| MATF | MID   |                     |       |       |       |      |      |       |    |
|      | “CRI” | Criterion<br>(=101) | MES   | DT    |       |      |      |       |    |
|      | “CRI” | Criterion<br>(=102) | D1    | D2    | D3    | D4   | D5   | ε0DOT |    |
|      |       |                     | TROOM | TMELT | CP    | MTH  |      |       |    |
|      | “CRI” | Criterion<br>(=103) | MPS   | MPS-C | VOLF  | DT   |      |       |    |
|      | “CRI” | Criterion<br>(=104) | PRS   |       |       |      |      |       |    |
|      | “CRI” | Criterion<br>(=105) | ALPHA | THETA | GAMMA | BETA | R    | D     |    |
|      |       |                     | W     | X0    | CBAR  | N    | TYPE | ITER  |    |
|      |       |                     | TOFF  |       |       |      |      |       |    |

**Example:**

|      |     |     |     |        |  |  |  |  |  |
|------|-----|-----|-----|--------|--|--|--|--|--|
| MATF | 7   |     |     |        |  |  |  |  |  |
|      | CRI | 101 | 2.0 | 1.0E20 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of MAT1 or MATORT entry. (Integer > 0; required.)  |
| “CRI”      | Enter the character string “CRI” to start input data for a failure criterion.  |
| Criterion  | Select an integer corresponding to the failure criterion to be applied. Only one failure criterion is allowed. (Integer; no Default) |
| 101        | Maximum Equivalent Stress Failure Model. It is only available for solid elements.  |
| 102        | Johnson-Cook Failure Model. It is only available for solid elements. See Remarks 1.  |
| 103        | Maximum Plastic Strain Failure Model. It is only available for Hugh-Liu beams, shells and solid elements.                            |
| 104        | Maximum Pressure Failure Model. It is only available for solid elements.   |
| 105        | CAP Failure Model. It is only acceptable for solid elements. See Remark 2.-4.  |

| Descriptor            | Meaning   |      |   |      |                                      |
|-----------------------|---|------|---|------|--------------------------------------|
| MES                   | Maximum equivalent stress that causes failure on the deviatoric part of the stress tensor. (Real > 0.0; required)   |      |   |      |                                      |
| DT                    | Minimum time step that causes total failure. (Real > 0.0; default=not used.)  |      |   |      |                                      |
| D1~D5                 | Damage coefficients. (Real; default=0.0)  |      |   |      |                                      |
| $\dot{\epsilon}$ 0DOT | Reference plastic strain rate. (Real; default=1.0)  |      |   |      |                                      |
| TROOM                 | Room temperature. (Real; default=0.0)   |      |   |      |                                      |
| TMELT                 | Melt temperature. (Real; default=1.0E20)  |      |   |      |                                      |
| CP                    | Heat capacity. (Real; default=1.0E20)   |      |   |      |                                      |
| MTH                   | Specifies how failure is applied. (Character; default=CONT) <table> <tr> <td>CONT</td><td>Continuous failure</td></tr> <tr> <td>DISC</td><td>Discrete failure</td></tr> </table>                              | CONT | Continuous failure                          | DISC | Discrete failure                     |
| CONT                  | Continuous failure  |      |   |      |                                      |
| DISC                  | Discrete failure  |      |   |      |                                      |
| MPS                   | Maximum plastic strain that causes failure. (Real > 0.0; required)  |      |   |      |                                      |
| MPS-C                 | Maximum plastic strain when material is under compression that causes failure. (Real > 0.0; default=MPS-C)  |      |   |      |                                      |
| VOLF                  | Minimum volumetric failure criteria. It is only available for solid elements. (Real > 0.0; default=1.0E-12)   |      |   |      |                                      |
| DT                    | Minimum time step that causes total failure. This option is only acceptable in solid elements. (Real > 0.0; default=not used)   |      |   |      |                                      |
| PRS                   | Maximum pressure that causes failure. (Real > 0.0; required)  |      |   |      |                                      |
| ALPHA                 | Failure envelope parameters. (Real $\geq$ 0.0; required)  |      |   |      |                                      |
| THETA                 | Failure envelope linear coefficient. (Real $\geq$ 0.0; required)  |      |   |      |                                      |
| GAMMA                 | Failure envelope exponential coefficient. See Remark 2. (Real $\geq$ 0.0; required)   |      |   |      |                                      |
| BETA                  | Failure envelope exponent. (Real $\geq$ 0.0; required)  |      |   |      |                                      |
| R                     | Cap surface axis ratio. (Real $\geq$ 0.0; required)   |      |   |      |                                      |
| D                     | Hardening law exponent. (Real $\geq$ 0.0; required)   |      |   |      |                                      |
| W                     | Hardening law coefficient. (Real $\geq$ 0.0; required)  |      |   |      |                                      |
| X0                    | Hardening law exponent. (Real $\geq$ 0.0; default=0.0)  |      |   |      |                                      |
| CBAR                  | Kinematic hardening coefficient. (Real $\geq$ 0.0; default=0.0)   |      |   |      |                                      |
| N                     | Kinematic hardening parameter. (Real $\geq$ 0.0; default=0.0)   |      |   |      |                                      |
| TYPE                  | Formulation type. (Character; default=SOIL) <table> <tr> <td>SOIL</td><td>Soil or concrete (cap surface may contract)</td></tr> <tr> <td>ROCK</td><td>Rock (cap surface does not contract)</td></tr> </table> | SOIL | Soil or concrete (cap surface may contract) | ROCK | Rock (cap surface does not contract) |
| SOIL                  | Soil or concrete (cap surface may contract)   |      |   |      |                                      |
| ROCK                  | Rock (cap surface does not contract)  |      |   |      |                                      |
| ITER                  | Iteration scheme. (Character; default=VEC) <table> <tr> <td>VEC</td><td>Fixed number of iterations (vectorized)</td></tr> </table>  | VEC  | Fixed number of iterations (vectorized)     |      |                                      |
| VEC                   | Fixed number of iterations (vectorized)   |      |   |      |                                      |

| Descriptor | Meaning  |
|------------|--|
| FULL       | Fully iterative  |
| TOFF       | Tension cut off (positive in compression). See Remark 3. (Real < 0.0; default=blank) |

**Remarks:**

- Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{\text{time}} \frac{\Delta \varepsilon_p}{\dot{\varepsilon}^{\text{frac}}}$$

$$\dot{\varepsilon}^{\text{frac}} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left( 1 + D_4 \ln \frac{\dot{\varepsilon}_{\text{pl}}}{\dot{\varepsilon}_{\text{pl}}^0} \right) (1 + D_5 T^*)$$

$$T = \text{SIE/CP}$$

$$T^* = \frac{T - T_{\text{room}}}{T_{\text{melt}} - T_{\text{room}}}$$

$$\sigma^* = \frac{\sigma_m}{\sigma}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature,  $\sigma_m$  is the mean stress,  $\sigma$  is the von Mises equivalent stress, and  $\dot{\varepsilon}^{\text{frac}}$  is the fracture strain.

The fracture strain depends on a nondimensional plastic strain rate,  $\dot{\varepsilon}_{\text{pl}} / \dot{\varepsilon}_{\text{pl}}^0$ . If D exceeds one it set equal to one.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

- For a physically meaningful model, the value of the failure envelope exponential coefficient should be less than the failure envelope parameter ( $\text{ALPHA} < \text{GAMMA}$ ).

3. The tension cut off value (TOFF) can be defined on the entry and must be less than zero. If the tension cut off is left blank, Nastran SOL700 calculates the tension cut off as the intersection point of the failure envelope surface with the J1-axis as described in Remark 4.
4. The cap material model can be used for geomechanical problems with materials like soil, concrete and rock.

The cap model is characterized by the following constitutive equations:

$$\varepsilon = \varepsilon^e + \varepsilon^p$$

$$\sigma = C(\varepsilon - \varepsilon^p)$$

where  $\varepsilon$ ,  $\varepsilon^e$  and  $\varepsilon^p$  are the total, elastic and plastic strain tensor,  $C$  is the elasticity matrix and  $\sigma$  is the stress tensor. The flow rule is given by:

$$\dot{\varepsilon}^p = \sum_{k=1}^e \dot{\lambda}_k \frac{\partial f_k}{\partial \sigma}$$

where the sum is over the active yield surfaces  $f^k$ , i.e., the failure envelope ( $k=1$ ), the hardening cap surface ( $k=2$ ), and the fixed tension cutoff surface ( $k=3$ ). The yield conditions are defined by:

$$f_1(\sigma) \leq 0$$

$$f_2(\sigma, \kappa) \leq 0$$

$$f_3(\sigma) \leq 0$$

The hardening parameter,  $\kappa$ , for the cap model is related to the plastic volume change by a hardening law.

The cap model is a plasticity model described by a yield surface that is defined by means of a failure envelope, a hardening cap and a tension cut off. Figure 1 shows the typical yield surfaces in a cap model.

The failure envelope surface is denoted by

$$f_1 = \sqrt{J_{2D}} - \text{MIN}(F_e(J_1), T_{mises})$$

and the cap by

$$f_2 = \sqrt{J_{2D}} - F_c(J_1, \kappa) \text{ for } L(\kappa) \geq J_1 \geq X(\kappa)$$

where  $J_1$  is the first invariant (trace) of the stress tensor,  $J_{2D}$  is the second invariant of the stress deviator,  $\kappa$  is an internal state variable that measures hardening as a functional of the history of plastic volumetric strain and  $L(\kappa)$  and  $X(\kappa)$  define the range of the cap. Note that  $J_1$  is chosen as negative in tension.

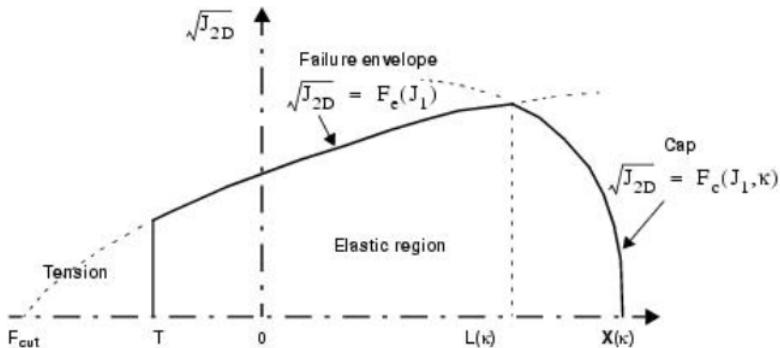


Figure 9-109      Typical Yield Surfaces in a Cap Model

The functions  $F_e$  and  $F_c$  are given by (see References 1. and 2.):

$$F_e(J_1) = a - \gamma \exp(-\beta J_1) + \theta J_1$$

$$F_c(J_1, \kappa) = \frac{1}{R} \sqrt{[X(\kappa) - L(\kappa)]^2 - [J_1 - L(\kappa)]^2}$$

The von Mises type transition failure surface is defined by the following:

$$T_{\text{mises}} = \frac{1}{R} |X(\kappa) - L(\kappa)|$$

The intersection of the cap with the (hydrostatic)  $J_1$  axis is given by:

$$X(\kappa) = \kappa + R F_e(\kappa)$$

and  $L(\cdot)$  is defined by:

$$L(\kappa) = \kappa = \begin{cases} \kappa & \text{if } \kappa > 0 \\ 0 & \text{if } \kappa \leq 0 \end{cases}$$

The hardening parameter,  $\kappa$ , is related to the actual plastic volume change:

$$\varepsilon_v^p(X) = W \{ 1 - \exp[-D(X(\kappa) - X_0)] \}$$

The tension cut off surface is given by the function:

$$f_3 = T - J_1$$

where  $T$  is the maximum hydrostatic tension sustainable by the material.

Kinematic work hardening for the failure envelope surface is based on the approach of Isenberg et. Al. [1978]. It is switched on by specifying  $N$ . The failure envelope surface is replaced by a family of envelope surfaces that are bounded by an initial yield surface and by a limiting failure envelope surface. Which member of the family

is taken, is implemented by replacing in all yield relations the stress tensor  $\sigma$  by  $\sigma - \zeta$  where  $\zeta$  is a deviatoric tensor that accumulates in time. This tensor  $\zeta$  is called the “back stress tensor” and is defined by

$$\frac{d\zeta}{dt} = \bar{c} \bar{F}(\sigma, \zeta) \frac{d\varepsilon^p}{dt}$$

$$F = \text{MAX}\left(0, 1 - \frac{(\sigma - \zeta) \cdot \zeta}{2NF_e(J_1)}\right)$$

Here  $\varepsilon^p$  is the deviatoric plastic strain tensor, N denotes the size of the yield surface and represents the radial distance between the outside of the initial yield surface and the inside of the limit surface. After each increment of  $\varepsilon^p$ , it is checked whether its second invariant exceeds N. In that case,  $\varepsilon^p$  is scaled by a scalar such that its second invariant equals N. For consistency between the limit surface of the kinematic hardening cap model and the failure envelope of the standard cap model, the parameter  $\bar{c}$  is placed by  $\bar{c} = N$ .

### References:

1. Sandler, I. S. and Rubin, D., “An Algorithm and a Modular Subroutine for the Cap Model,” International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 3, 173-186 (1979)
2. Simo, J. C., Ju, J.W., Pister, K. S., and Taylor, R.L., “Assessment of Cap Model: Consistent Return Algorithms and Rate-dependent Extension,” Journal of Engineering Mechanics, Vol. 114, No. 2, February 1988.
3. Isenberg, Jeremy, D K. Vaughan, and I Sandler. Nonlinear Soil-Structure Interaction. Palo Alto, Calif: The Institute, 1978. Print.

**MATF1****Frequency Dependent Isotropic Material Definition**

Defines the frequency dependent properties for an isotropic poroelastic material.

**Format:**

| 1     | 2     | 3     | 4     | 5     | 6      | 7    | 8 | 9     | 10 |
|-------|-------|-------|-------|-------|--------|------|---|-------|----|
| MATF1 | MID   | T(E)  | T(G)  | T(NU) | T(RHO) | T(A) |   | T(GE) |    |
|       | T(ST) | T(SC) | T(SS) |       |        |      |   |       |    |

**Example:**

|       |   |    |  |  |     |  |  |  |  |
|-------|---|----|--|--|-----|--|--|--|--|
| MATF1 | 1 | 30 |  |  | 130 |  |  |  |  |
|-------|---|----|--|--|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Material property identification number that matches the identification number on MAT1 entry. (Integer $\geq 0$ ; Required) |
| T(E)       | Identification number of a TABLEMi entry for the Young's modulus. (Integer $\geq 0$ or Blank)                               |
| T(G)       | Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or Blank)                                 |
| T(NU)      | Identification number of a TABLEMi entry for the Poisson's ratio. (Integer $\geq 0$ or Blank)                               |
| T(RHO)     | Identification number of a TABLEMi entry for the mass density. (Integer $\geq 0$ or Blank)                                  |
| T(A)       | Identification number of a TABLEMi entry for the thermal expansion coefficient. (Integer $\geq 0$ or Blank)                 |
| T(GE)      | Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or Blank)                           |
| T(ST)      | Identification number of a TABLEMi entry for the tension stress limit. (Integer $\geq 0$ or Blank)                          |
| T(SC)      | Identification number of a TABLEMi entry for the compression stress limit. (Integer $\geq 0$ or Blank)                      |
| T(SS)      | Identification number of a TABLEMi entry for the shear stress limit. (Integer $\geq 0$ or Blank)                            |

**Remarks:**

1. Remarks 1 to 4 under [MATT1, 2375](#) are applicable with MATF1..
2. MATF1 must reside under Case Control 'BEGIN BULK TRMC' and is applicable to that trim component only. It may only be referenced by MATPE1 entry for a poroelastic element.

**MATFAB****Woven Fabric Material- SOL700**

Defines the properties of a bi-directional woven fabric material for shell elements. Used in SOL700 only.

**Format:**

| 1      | 2    | 3   | 4       | 5       | 6        | 7        | 8        | 9    | 10 |
|--------|------|-----|---------|---------|----------|----------|----------|------|----|
| MATFAB | MID  | RHO | ECOAT   | NUCOAT  | GCOAT    | DAMPCOAT | COMPRESS | PERC |    |
| +      | E1L  | E1Q |         | THETA1  | XWARP    | YWARP    | ZWARP    |      |    |
| +      | E2L  | E2Q |         | THETA2  | XWEFT    | YWEFT    | ZWEFT    |      |    |
| +      | SCOF | G12 | DAMPFIB | COMPFIB | LOCKANG1 | LOCKANG2 |          |      |    |

**Example:**

|        |        |     |        |      |     |     |     |  |  |
|--------|--------|-----|--------|------|-----|-----|-----|--|--|
| MATFAB | 3      | 850 | 5.52E6 | 0.33 |     |     |     |  |  |
| +      | 21.6E7 |     |        |      | 1.0 | 0.0 | 0.0 |  |  |
| +      | 21.6E7 |     |        |      | 0.0 | 1.0 | 0.0 |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Unique material number. (Integer > 0; required)  |
| RHO        | Density. (Real > 0.0; required)  |
| ECOAT      | Young's modulus of coating material. See Remark 1. (Real > 0.0; default=blank)   |
| NUCOAT     | Poisson's ratio of coating material. See Remark 1. (Real > 0.0; default=blank)   |
| GCOAT      | Shear modulus of coating material. See Remark 1. (Real > 0.0; default=blank)   |
| DAMPCOAT   | Damping is applied to the coating stresses. See Remark 2. (Real >0.0; default=0.1)   |
|            | $d\sigma_{ij} = DAMPCOAT \cdot ECOAT \cdot \epsilon_{ij} dt_{elm}$   |
| COMPRESS   | Scale factor for coating compression stresses. See Remark 3. (0.0? Real?1.0; default=1.0)  |
| PERC       | Thickness percentage of coating material. (0.0? Real?100.0; default=0.0=no coating)  |
| E1L        | Young's modulus of fabric in warp direction, linear coefficient. (Real > 0.0; required)  |
| E1Q        | Young's modulus of fabric in warp direction, quadratic coefficient. (Real >0.0; default=0.0)   |
| THETA1     | Orientation angle between the element coordinate system and the warp ends. See Remark 4. (Real; default=blank)   |
| XWARP      | X component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0) |
| YWARP      | Y component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0) |

| Descriptor | Meaning  |
|------------|--|
| ZWARP      | Z component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0) |
| E2L        | Young's modulus of fabric in weft direction, linear coefficient. (Real > 0.0; required)  |
| E2Q        | Young's modulus of fabric in weft direction, quadratic coefficient. (Real ? 0.0; default=0.0)  |
| THETA2     | Orientation angle between the element coordinate system and the weft ends. See Remark 4. (Real; default=blank)   |
| XWEFT      | X component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0) |
| YWEFT      | Y component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0) |
| ZWEFT      | Z component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0) |
| SCOF       | Shear coefficient of friction. See Remark 6. (Real; default=0.0)   |
| G12        | Shear modulus of fabric material. See Remark 7. (Real; default=blank)  |
| DAMPFIB    | Damping is applied to the fiber stresses. See Remark 2. (Real $\geq$ 0.0; default=0.1)   |
|            | $d\sigma_{ij} = DAMPCOA \bullet E\varepsilon_{ij}dt_{elm}$   |
| COMPFB     | Scale factor for fiber compression stresses. See Remark 3. ( $0.0 \leq$ Real $\leq$ 1.0; default=1.0)  |
| LOCKANG1   | Locking angle 1 for change in fiber cross-over angle. See Remark 10. (Real $\geq$ 0.0; default=10.0)   |
| LOCKANG2   | Locking angle 2 for change in fiber cross-over angle. See Remark 10. (Real $\geq$ 0.0; default=15.0)   |

#### Remarks:

1. When a coating is defined (PERC>0), two out of three values need to be specified for ECOAT, NUCOAT, and GCOAT.
2. For air bag modeling the following values of DAMPCOAT and DAMPFIB are suggested:  
 $DAMPCOAT = 0.05$   
 $DAMPFIB = 0.05$
3. The compressive stresses in the fibers are scaled with the value of COMPFB. Putting COMPFB = 0.0 results in a tension only fiber model.

The compressive stresses in the coating are scaled with the value of COMPCOAT. Putting COMPCOAT = 0.0 results in a tension only coating model.

The compressive stresses are scaled in the direction of the principal stresses.

When PERC = 100%, and the coating of this fabric model is used to simulate an isotropic air bag material, it is best to scale down the compressive stresses of the coating. A suggested value is COMPCOAT = 0.1

4. Since this is a model which tracks warp and weft directions and uses total warp/weft strain as a state variable, the initial warp and weft directions must be specified. There are two ways to indicate the initial warp and weft directions:

- a. THETA1 and THETA2

Orientation angles between the element coordinate system and the warp/weft ends. If no orientation angle is specified, vectors will be used to indicate the warp/weft directions of the fabric material with respect to the basic coordinate system.

- b. XWARP, YWARP, ZWARP and XWEFT, YWEFT, ZWEFT

Vectors indicating the warp/weft directions of the fabric material with respect to the basic coordinate system. The projection of these vectors on the surface of each element is used to determine the angle between the element and the material coordinate system. If the orientation angles are defined, these vectors are ignored.

5. For shell element properties (PSHELL1), when the material is MATFAB, the material angle THETA is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.

For layered composite element properties (PCOMP), when the material of a ply is MATFAB, the angle THETAi is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.

6. The maximum shear stress is given by a friction coefficient of the fabric (SCOF) times the RMS value of the direct fiber stresses.

7. If the field G12 is left blank, the shear modulus is computed from the RMS value of the two linear stiffness coefficients.

8. When MATFAB material is referenced by shell elements, the Spin Rate method (SPIN) is applied automatically when no stress-rotation correction is specified on SPINCOR option. See PSHELL1 entry for the details on SPINCOR option.

9. There are a number of specific output sublayer variables useful for this material:

Q1AF Direction cosines/sines between the element coordinate

Q2AFIB System and the warp ends

Q1BFIB Direction cosines/sines between the element coordinate

Q2BFIB System and the weft picks

SGMA Direct stress in fabric parallel to the warp ends

SGMB Direct stress in fabric parallel to the weft picks

SGFRIC Stress due only to shear in the weave of the fabric

EPSFA Strain in fabric parallel to the warp ends

EPSFB Strain in fabric parallel to the weft picks

ANGLE Crossover angle between warp ends and weft picks

10. When a fabric is being sheared, the angle between the fibers changes. At a certain moment, the fibers will reach a locking angle, after which a further change in the fiber angle is no longer possible.

The simulation models this behavior as follows:

- a. Change in Fiber Crossover Angle < LockAng1

The shear stress between the fibers is cut off based on the friction coefficient SCOF

- b. LockAng1 < Change in Fiber Crossover Angle < LockAng2

The shear stress between the fibers is linearly increased.

- c. Change in Fiber Crossover Angle > LockAng2

The shear stress between the fibers is no longer cut off.

This situation is equal to an infinite friction coefficient SCOF.

**MATFTG****Fatigue Material Properties**

Defines fatigue material properties for time domain based SOLs 101, 103, 112 and frequency domain based SOLs 108 and 111.

**Format (SOL 101, 103, 112):**

| 1      | 2        | 3      | 4      | 5      | 6      | 7      | 8      | 9       | 10 |
|--------|----------|--------|--------|--------|--------|--------|--------|---------|----|
| MATFTG | MID      | CNVRT  |        |        |        |        |        |         |    |
|        | "STATIC" | YS     | UTS    | CODE   | TYPE   | RR     | SE     | mp      |    |
|        |          | RA     |        |        |        |        |        |         |    |
|        | "SN"     | SRI1   | b1     | Nc1    | b2     | Nfc    | SE     | BTHRESH |    |
|        | "SNS1"   | M1     | M2     | M3     | M4     | MSS    | RTHICK | nTHICK  |    |
|        | "SNS2"   | SF-FXY | DE-FXY | TE-FXY | SF-MXY | DE-MXY | TE-MXY |         |    |
|        | "SNBR1"  | SF-FZ  | DE-FZ  | TE-FZ  | SF-MZ  | DE-MZ  | TE-MZ  |         |    |
|        | "SNBR0"  |        |        |        |        |        |        |         |    |
|        | "TABLE"  | VALUE1 | TID1   | VALUE2 | TID2   | VALUE3 | TID3   |         |    |
|        |          | VALUE4 | TID4   | ...    | ...    | -etc.- |        |         |    |
|        | "BASTEN" | A      | B      | c      | Eb     | Sd     |        |         |    |
|        | "EN"     | Sf     | b      | c      | Ef     | n'     | K'     | Nc      |    |
|        |          | SEe    | SEp    | SEc    | Ne     | FSN    | S      |         |    |
|        | "MATID"  | E      | NU     |        |        |        |        |         |    |
|        |          | MID1   | MID2   | MID3   | MID4   | MID5   | MID6   | MID7    |    |
|        |          | -etc.- |        |        |        |        |        |         |    |

**Format (SOL 108, 111):**

| 1      | 2        | 3      | 4      | 5    | 6    | 7    | 8    | 9     | 10 |
|--------|----------|--------|--------|------|------|------|------|-------|----|
| MATFTG | MID      | CNVRT  | MATOUT |      |      |      |      |       |    |
|        | "STATIC" | YS     | UTS    |      | TYPE |      |      |       |    |
|        | "SN"     | SRI1   | b1     | Nc1  | b2   | Nfc  |      |       |    |
|        | "TABLE"  | VALUE1 | TID1   |      |      |      |      |       |    |
|        | "MMPDS"  | A1     | A2     | A3   | A4   |      |      |       |    |
|        | "EN"     | Sf     | b      | c    | Ef   | n'   | K'   | GAMMA |    |
|        |          | SEe    | SEp    | SEc  | Ne   |      |      | TOLER |    |
|        | "MATID"  | E      | NU     |      |      |      |      |       |    |
|        |          | MID1   | MID2   | MID3 | MID4 | MID5 | MID6 | MID7  |    |
|        |          | -etc.- |        |      |      |      |      |       |    |

**Examples:**

|        |        |     |     |    |  |  |  |  |  |
|--------|--------|-----|-----|----|--|--|--|--|--|
| MATFTG | 9      |     |     |    |  |  |  |  |  |
|        | STATIC | 430 | 682 | 99 |  |  |  |  |  |

|        |        |      |         |      |     |      |  |     |  |
|--------|--------|------|---------|------|-----|------|--|-----|--|
| MATFTG | 9      |      |         |      |     |      |  |     |  |
|        | STATIC | 430  | 682     |      |     |      |  | 0.1 |  |
|        | SN     | 3095 | -0.1339 | 1.e8 | 0.0 | 1.e8 |  |     |  |

|        |        |       |      |    |    |  |  |     |  |
|--------|--------|-------|------|----|----|--|--|-----|--|
| MATFTG | 9      |       |      |    |    |  |  |     |  |
|        | STATIC | 430   | 682  |    |    |  |  | 0.1 |  |
|        | MATID  | 2.1e5 | 0.3  |    |    |  |  |     |  |
|        |        | 9     | THRU | 15 | 18 |  |  |     |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Unique material ID that matches the identification of a MAT1 entry unless the MATID line is supplied (Integer>0). See Remark 1.                          |
| CNVRT      | Conversion factor. See Remark 2. regarding units.  |
| MATOU<br>T | SOL 108, 111 only. This is a debug option for writing the fully derived material curve to a file (Integer 0 or 1; Default = 0).                          |
| STATIC     | Required flag indicating that yield and/or ultimate tensile strengths and other common parameters are supplied (Character=STATIC). See Remark 3. and 4.  |
| YS         | Yield strength (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 3000$ ). See Remark 2. regarding units.                                       |
| UTS        | Ultimate tensile strength (Valid range equivalent in MPa: $100.0 \leq \text{Real} \leq 4000$ ). See Remark 2. regarding units.                           |
| CODE       | SOL 101, 103, 112 only. Material code used in automatically generating S-N or $\epsilon$ -N data curves and for surface finish correction. See Remark 3. |
| TYPE       | SOL 101, 103, 112: Specification of the type of S-N curves defined using the TABLE or BASTEN flag (no Default). See Remark 4.                            |
|            | SOL 108, 111: Specification as to whether material definitions are specified as RANGE or AMPLitude. See Remark 4.  |
| RR         | SOL 101, 103, 112 only. R-ratio of test. ( $-1.0e30 \leq \text{Real} \leq 1.0$ , Default=-1.0).  |
| SE         | SOL 101, 103, 112 only. Standard Error of Log(N). ( $0.0 \leq \text{Real} \leq 10.0$ , Default=0.1).   |
| mp         | SOL 101, 103, 112 only. Plastic Poisson's Ratio. ( $0.45 \leq \text{Real} \leq 0.55$ , Default=0.5).   |

| Descriptor  | Meaning   |
|-------------|---|
| RA          | <b>SOL 101, 103, 112 only.</b> Percentage reduction in area used for deriving $\varepsilon$ -N data for metals of type "other" (TYPE field = 200-207 or >300). (0.0 < Real ≤ 100.0; no Default).  |
| SN          | Flag indicating the definition of an S-N curve(s) follow (Character = SN, optional). See Remark 5. and 7. and 8.  |
| SNS1/SNS2   |   |
| SNBR1/SNBR0 |   |
| SRI1        | Stress range intercept. (Valid range equivalent in MPa: 1.0 ≤ Real ≤ 2.5e4, no Default). See Remark 2 regarding units.  |
| b1          | First fatigue strength exponent. (-1.0 < Real < -0.02, no Default)  |
| Nc1         | In 1-segment S-N curve, the cycles limit of endurance. In 2-segment S-N curve, this is the fatigue transition point. Both are defined in cycles. (1.0 ≤ Real ≤ 1.0E25; no Default).   |
| b2          | Second fatigue strength exponent. It is zero when defining 1-segment S-N curve; (-0.5 < Real ≤ 0.0, Default=0.0).   |
| Nfc         | Fatigue cutoff. (1.e-9 ≤ Real ≤ 1.0e30; Default=1.0e30).  |
| SE          | <b>SOL 101, 103, 112 only.</b> Standard Error of Log(N). (0.0 ≤ Real ≤ 10.0, Default=0.1). SE from "STATIC" line will be used if left blank.  |
| BTHRESH     | <b>SOL 101, 103, 112 only.</b> Threshold value of the bending (r) ratio used in interpolation between "stiff" and "flexible" SN curves for fatigue analysis of seam welds. (0.0 ≤ Real ≤ 0.999; Default=0.5)  |
| M1 - M4     | <b>SOL 101, 103, 112 only.</b> Mean stress slope parameters M1 through M4 representing sensitivity to mean stress in four (4) regimes of R-ratio as plotted on a constant life Haigh diagram and used in FKM mean stress correction. (-0.99 ≤ Real ≤ 0.0; Default = blank). See Remark 6.             |
| MSS         | <b>SOL 101, 103, 112 only.</b> Mean stress sensitivity factor, used only for fatigue analysis of spot welds with CORR=SIMPLE on the FTGPARM entry. (0.0 ≤ Real ≤ 1.0, Default=blank). This mean stress correction method is a simplified version of the FKM method where M1=M2=M3=M4=-MSS.            |
| RTHICK      | <b>SOL 101, 103, 112 only.</b> Reference thickness/threshold (in consistent model length units) for sheet thickness correction used in fatigue analysis of seam welds. Ignored if THICK=0 on the <a href="#">FTGPARM</a> entry for "SEAMW". Must be supplied if THICK=1. (Real ≥ 1.0e-9; Default=1.0) |
| nTHICK      | <b>SOL 101, 103, 112 only.</b> Sheet thickness correction exponent used in the fatigue analysis of seam welds. Ignored if THICK=0 on the <a href="#">FTGPARM</a> entry for "SEAMW". Must be supplied if THICK=1. (Real; Default=0.16667)  |
| SF-FXY      | <b>SOL 101, 103, 112 only.</b> Scale factor for stress due to FX or FY (shear forces),  |
| SF-MXY      | MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.   |
| SF-FZ       |   |
| SF-MZ       |   |

| Descriptor | Meaning   |
|------------|---|
| DE-FXY     | <b>SOL 101, 103, 112</b> only. Diameter exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.   |
| DE-MXY     |   |
| DE-FZ      |   |
| DE-MZ      |   |
| TE-FXY     | <b>SOL 101, 103, 112</b> only. Thickness exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.  |
| TE-MXY     |   |
| TE-FZ      |   |
| TE-MZ      |   |
| TABLE      | Flag indicating the definition of S-N curves as a number of tables follows (Character=TABLE; optional). See Remark 5.   |
| VALUEi     | <b>SOL 101, 103, 112</b> : The constant mean stress, R-ratio, or life (in cycles) of this particular S-N curve. (Real; no Default). See Remark 2 regarding units.<br><br><b>SOL 108, 111</b> : This should be specified as 1.0 as only one curve is supported and should be defined as zero mean.   |
| TIDi       | <b>SOL 101, 103, 112</b> : A TABLEM1 ID defining the S-N curve of stress range (y) vs. life (x) (in cycles) for this particular S-N curve, or mean stress (x) vs. stress amplitude (y) for constant life (Haigh diagram) curves (Integer>0). See Remark 2. regarding units.<br><br><b>SOL 108, 111</b> : A TABLEM1 (may cause unintended recalculation of eigenvalues if used in RESTART job) or TABRND1 ID defining the S-N curve of stress (y) vs. life (x) (in cycles). Only one curve is allowed.and should be defined at zero mean. See Remark 2. regarding units. |
| MMPDS      | <b>SOL 108, 111</b> only. Flag indicating the definition of an S-N curve based on the MMPDS S-N approach (Character-MMPDS; optional). See Remark 5.   |
| A1         | Coefficients or exponent of the standard MMPDS-01 material equation (must be defined in units-KSI based on amplitude). (Real; typical ranges  |
| A2         | $5 \leq A1 \leq 110$ ; $-47 \leq A2 \leq -1.0$ , $0.1 \leq A3 \leq 1.0$ ; $0.0 \leq A4 \leq 114$ ; no Default).   |
| A3         |   |
| A4         | CNVRT is not applied to these values at all - it is ignored!  |
| BASTEN     | <b>SOL 101, 103, 112</b> only. Flag indicating the definition of an S-N curve based on the Bastenaire S-N approach (Character-BASTEN; optional). See Remark 5.  |
| A          | Bastenaire coefficient - a parameter positioning the curve along the life axis. (Valid range equivalent in MPa: $1.0e4 \leq \text{Real} \leq 1.0e10$ ). See Remark 2. regarding units.  |
| B          | Scale Factor parameter. (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$ ). See Remark 2. regarding units.  |
| c          | Bastenaire exponent. ( $0.01 \leq \text{Real} \leq 1.0e10$ )  |
| Eb         | Bastenaire fatigue limit (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$ ). See Remark 2 regarding units.  |

| Descriptor | Meaning  |
|------------|--|
| Sd         | Bastenaire scatter factor. (Valid range equivalent in MPa: $0.0 \leq \text{Real} \leq 1.0e25$ ). See Remark 2 regarding units.   |
| EN         | Flag indicating the definition of an $\varepsilon$ -N curve is to follow. (Character=EN; optional). See Remark 5..   |
| Sf         | Fatigue strength coefficient (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 4000.0$ ; no Default). See Remark 2. regarding units.                                   |
| b          | Fatigue strength exponent ( $-0.5 \leq \text{Real} \leq -2.0e-3$ ; no Default).  |
| c          | Fatigue ductility exponent. ( $-1.2 \leq \text{Real} \leq -0.15$ ; no Default).  |
| Ef         | Fatigue ductility coefficient, ( $0.001 \leq \text{Real} \leq 10.0$ ; no Default).   |
| n'         | Cyclic strain-hardening exponent, ( $5.0e-3 \leq \text{Real} \leq 0.5$ ; no Default).  |
| K'         | Cyclic strength coefficient, (Valid range equivalent in MPa: $50 \leq \text{Real} \leq 1.2e4$ ); no Default). See Remark 5. regarding units.                                     |
| Nc         | SOL 101, 103, 112 only. Cut-off (reversals), ( $1.0e5 \leq \text{Real} \leq 1.0e25$ ; Default= $2.0e8$ ).  |
| GAMMA      | SOL 108, 111 only. Additional coefficient needed for the Walker equation, ( $0.0 \leq \text{Real} \leq 1.0$ ; Default=0.0).  |
| SEe        | SOL 101, 103, 112 only. Standard Error of Log(e) (elastic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0).   |
| SEp        | SOL 101, 103, 112 only. Standard Error of Log(e) (plastic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0 ).  |
| SEC        | SOL 101, 103, 112 only. Standard Error of Log(e) (cyclic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0).  |
| Ne         | SOL 101, 103, 112 only. Endurance Limit (Reversals), ( $1.0 e-9 \leq \text{Real} \leq 1.0e25$ ; Default= $2.0e8$ ).  |
| FSN        | SOL 101, 103, 112 only. Fatemi-Socie Parameter, ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.6)  |
| S          | SOL 101, 103, 112 only. Wang Brown Parameter, ( $-10.0 \leq \text{Real} \leq 10.0$ ; Default=1.0)  |
| TOLER      | SOL 108, 111 only. Parameter used for frequency based strain life analysis with Neuber notch correction in vibration fatigue only. ( $\text{Real} >= 1e-10$ , Default = $1e-5$ ) |
| MATID      | Flag indicating that a list of MAT1 IDs is to follow. See Remark 1.  |
| E          | Young's modulus, (Valid range equivalent in MPa: $\text{Real} > 0.0$ , no Default).  |
| NU         | Elastic Poisson's ratio. ( $0.25 \leq \text{Real} \leq 0.35$ , Default = 0.3)  |
| MIDI       | IDs of existing MAT1 entries to which this MATFTG entry will be linked. At least one ID must be supplied. The "THRU" keyword is supported.                                       |

**Remarks:**

1. Element properties must reference MAT1 entries in order to be linked to a MATFTG entry as only metal fatigue analysis of isotropic materials is supported. The MID must match that of an existing MAT1 entry called out by the property entry (e.g. PSHELL) unless the "MATID" line is provided, in which case all the MIDI referenced on the "MATID" line are then linked to the MATFTG entry. When the "MATID" line is used, E (required) and NU (optional) must be provided on this line as the code has no way of knowing from which referenced MAT1 this data should be extracted. Young's Modulus (E) and elastic Poisson's ratio (NU) are extracted from the corresponding MAT1 entry of the same ID when no "MATID" line is present. Usage of E and NU in the fatigue analysis itself is dependent on the type of fatigue analysis requested and fatigue material properties provided. Young's Modulus (E) is mainly used for deriving S-N or  $\epsilon$ -N data when none is provided. If the "MATID" line is used, and a MAT1 of the same ID also exists, that MAT1 ID must be in the MIDI list or it will be ignored in the analysis. In other words, when the "MATID" line is used, only the supplied MIDI IDs are used in the fatigue analysis.
2. The CNVRT field is only used if fatigue material stress based parameters are directly input using the "STATIC," "SN," "EN," "TABLE," or "BASTEN" methods. It is used to allow the user to input the fatigue material stress related parameters, (YS, UTS, SRI1, VALUEi, TIDI, A, B, Eb, K) in different units other than the model's consistent units. Example: model is producing stresses in PSI units, fatigue material parameters input in MPa, the CNVRT factor should be 145.0377 to convert MPa to PSI. Note that the y-values of any referenced TABLEM1 entries are also converted for the S-N method or both the x- and y-values when TYPE=LIFE (Haigh curves). It is also necessary to use the DTI,UNITS for defining the model's stress units. See DTI,UNITS. (Real; Default=1.0)
3. For SOLs 101, 103, and 112, if only STATIC is supplied, then for standard S-N or  $\epsilon$ -N analysis, the S-N or  $\epsilon$ -N curve is derived using the UTS and a material CODE. For fatigue analysis of spot or seam welds, a standard material set is used based on CODE, and UTS can be set to any valid number as the UTS from the standard material set will be used instead. Valid codes are listed in [Table 9-22](#) below. At a minimum UTS must be supplied along with the material CODE and E on MAT1, or an error will be issued. If either flag (SN or EN) is present, then the automatic generation is suppressed if all the data necessary to define S-N or  $\epsilon$ -N curves are given. For SN, these fields are SRI1, b1, Nc1. For EN, these fields are Sf, b, c, Ef, n', K'. Either all of them are present, in which case they are used directly, or all of them are omitted, in which case the parameters are derived as mentioned above. The determination as to whether S-N or  $\epsilon$ -N curves are generated is determined by the TYPE field set on the FTGPARM entry. If surface finish corrections are to be applied, CODE is also required (see PFTG entry). When curves are derived, the specified CODE gets internally converted to a generic code *ferrous*=99 for CODE<100, *aluminium*=100 for 100 ≤ CODE < 200, *other*=0 for 200 ≤ CODE < 300, or *titanium*=300 for 300 ≤ CODE < 400. When SN or EN flags exist, the generic code is set to *other*, but the CODE is used as given.
- For SOLs 108, and 111, if only STATIC is supplied, see Remark 5.
4. For SOLs 101, 103, and 111: TYPE can be set to AMPL, RANGE, or MAX for amplitude, range, or maximum stress, respectively, when using the "BASTEN" flag. Or TYPE can be set to MEAN (or AMEAN), RRATIO (or ARRATIO), or LIFE (or ALIFE) for constant mean, R-ratio, or life (Haigh diagrams), respectively, when using the "TABLE" flag. When using multiple S-N curves for mean

stress correction, CORR=INTERP on FTGPARM must be used. In this case, for RRATIO, a curve at R=-1 is required and for MEAN, a curve at zero (0) mean is required. AMPL, RANGE, and MAX are used to define stress types of Bastenaire models. MEAN vs. AMEAN, or RRATIO vs. ARRATIO, or LIFE vs. ALIFE indicates the stress type of the curves is range or amplitude, respectively.

**For SOLs 108 and 111:** TYPE can be set to AMPL or RANGE to indicate that the S-N curve is defined as amplitude or range and is applicable for the SN and TABLE lines.

5. **For SOLs 101, 103, and 111:** If an S-N analysis is specified (TYPE field on FTGPARM entry) but no S-N curve is supplied, an error will be issued. If an  $\varepsilon$ -N analysis is specified but no  $\varepsilon$ -N curve is supplied, an error will be issued. The STATIC data is required if SN or EN flags are specified on the MATFTG entry. The SN, BASTEN, and TABLE flags are generally mutually exclusive. When both SN and TABLE lines are defined, if TYPE is set, the SN parameters (SRI1, b1, Nc1, b2) will be ignored. EN flag may be present with SN, BASTEN, or TABLE flags, but one or the other will be ignored based on the type of analysis as set using the TYPE field on the FTGPARM entry. See Remark 7. and 8. for other “SNS1/SNS2/SNBR1/SNBR2” definitions.

**For SOLs 108 and 111:** The STATIC line is required and at least one of SN, TABLE, MMPDS, or EN lines. If no SN, TABLE, MMPDS, or EN lines are present, then the fatigue analysis is skipped and only the random vibration analysis is performed<sup>1</sup>. The use of the TYPE field in the FTGPARM entry has limited affect. Please note: If only SN data is present, it is used by default. If only EN data is present, it is used by default. The SN, TABLE, and MMPDS entries are mutually exclusive and if more are present for any particular MID, then the following hierarchical rule is implement to choose which to use: MMPDS, TABLE, SN. If both SN and EN are defined for a particular MID, EN is selected by default unless TYPE on the FTGPARM is set, in which case the corresponding SN or EN definition is used. However, TYPE is ignored if the corresponding material definition is not present, and the material definition used is then based on the hierarchy just defined.

6. **For SOLs 101, 103, and 111:** Mean stress sensitivity: M1 for R>1; M2 for  $-\infty < R < 0$ ; M3 for  $0 < R < 0.5$ ; M4 for  $0.5 < R < 1$ . If M<sub>1-4</sub> are undefined, and the material type (CODE) is given , all the parameters will be estimated using empirically defined rules for the FKM mean stress correction method. If only M<sub>2</sub> is defined, then M<sub>1</sub> and M<sub>4</sub> will be set to zero and M<sub>3</sub> to M<sub>2</sub>/3.
7. **For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of spot welds, “SNS1” and “SNS2” are used to define the SN curves of the top and bottom sheets to which the spot weld is connected and “SN” is used to define the SN curve of the weld nugget. If no “SN”, “SNS1”, or “SNS2” flags are present, generic SN parameters are used for steel or aluminum per the material CODE supplied. See [Table 9-23](#) for default values used in this case. If CODE is blank, steel is assumed. If only one of the flags is provided, then “SN” = “SNS1” = “SNS2”. If “SNS1” or “SNS2” is missing but the other is present, then “SNS1” = “SNS2”. If “SNS1” and “SNS2” are defined, but not “SN”, then “SN” = “SNS1”.
8. **For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of seam welds, “SNBR1” and “SNBR0” are used to define the flexible (bending ratio=1.0) and stiff (bending ratio=0.0) SN curves. If no “SN”, “SNBR1”, or “SNBR0” flags are present, generic SN parameters are used for steel or aluminum per the material

---

<sup>1</sup>If the EN line is supplied with only n' and K' defined, then the Neuber notch correction is also applied in the random only analysis affecting the resultant output stress PSD.

CODE supplied. See [Table 9-23](#) for default values used in this case. If CODE is blank, steel is assumed. If only "SN" is supplied, then the "SN" parameters are used for both "SNBR1" and SNBR0". If "SNBR1" or "SNBR0" is missing but the other is present, then "SNBR1" = "SNBR0". The 3rd and 4th lines of data for these keywords should not be entered as they are applicable to fatigue analysis of spot welds only, and not for seam welds.

9. For SOLs 101, 103, and 111: If SF, DE, & TE values are not supplied, default values are used based on material CODE for steel or aluminum. See [Table 9-24](#) and . These should only be entered for fatigue analysis of spot welds and not for anything else.

Table 9-22      Table of Material CODEs

| CODE | Description  |
|------|--|
| 1    | Flake cast iron (FCI)  |
| 2    | Ferritic cast iron with compacted graphite (FCICG)                           |
| 3    | Pearlitic cast iron with compacted graphite (PCICG)                          |
| 4    | Bainitic cast iron with compacted graphite (BCICG)                           |
| 5    | Ferritic cast iron with spheroidal graphite (FCISG)                          |
| 6    | Ferrite/pearlite cast iron with spheroidal graphite (FPCISG)                 |
| 7    | Pearlitic cast iron with spheroidal graphite (PCISG)                         |
| 8    | Bainitic cast iron with spheroidal graphite (BCISG)                          |
| 9    | Cast steel with less than 0.2% carbon (CSL2C)                                |
| 10   | Normalized cast steel with 0.2-0.4% carbon (NCS24C)                          |
| 11   | Quenched & tempered cast steel with 0.2-0.4% carbon (QTCS24)                 |
| 12   | Normalized cast steel with 0.4-0.7% carbon (NCS47)                           |
| 13   | Plain carbon wrought steel with < 0.2% carbon (PCWS)                         |
| 14   | Hot rolled/normalized plain carbon wrought steel, 0.2-0.4% carbon (HNPCWS24) |
| 15   | Quenched & tempered cast steel with 0.4-0.7% carbon (QTCS47)                 |
| 16   | Quenched & tempered plain carbon wrought steel, 0.2-0.4% carbon (QTPCWS24)   |
| 17   | Hot rolled/normalized plain carbon wrought steel, 0.4-0.7% carbon (HNPCWS47) |
| 18   | Quenched & tempered plain carbon wrought steel, 0.4-0.7% carbon (QTPCWS47)   |
| 19   | Normalized low alloy wrought steel (NLAWS)                                   |
| 20   | Quenched & tempered low alloy wrought steel (QTHSLAWS)                       |
| 21   | Normalized Ni/Cr/Mo wrought steel (NNCMWS)                                   |
| 22   | Quenched & tempered Ni/Cr/Mo wrought steel (QTNCMWS)                         |
| 23   | Austenitic stainless steel (ASS)   |
| 24   | Ferritic stainless steel (FSS)   |
| 25   | Martensitic stainless steel (MSS)  |

| CODE | Description  |
|------|--|
| 26   | Annealed plain carbon wrought steel, 0.2-0.4% carbon (APCWS24) |
| 27   | Annealed plain carbon wrought steel, 0.4-0.7% carbon (APCWS47) |
| 28   | Normalized carbon/manganese steel (MCMS)                       |
| 29   | Quenched and tempered carbon/manganese steel (QTCMS)           |
| 30   | Hardened chromium steel (HCS)                                  |
| 31   | Quenched and tempered chromium steel (QTCS)                    |
| 99   | Steel of unknown heat treatment (STEEL)                        |
| 100  | Wrought aluminium (WA)   |
| 101  | Wrought aluminium-copper alloy (WACA)                          |
| 102  | Wrought aluminium-manganese alloy (WAMNA)                      |
| 103  | Wrought aluminium-magnesium alloy (WAMGA)                      |
| 104  | Wrought aluminium-magnesium-silicon alloy (WAMGSA)             |
| 105  | Wrought aluminium-zinc alloy (WAZA)                            |
| 106  | Cast aluminium alloy (CAA)                                     |
| 107  | Wrought complex special purpose aluminum alloys (WCSPAA)       |
| 200  | Wrought copper (WCU)   |
| 201  | Wrought brass (WBR)  |
| 202  | Wrought aluminium bronze (WABR)                                |
| 203  | Cupronickel (CUPNI)  |
| 204  | Nickel silver (NIAG)   |
| 205  | Wrought phosphor bronze (WPHBR)                                |
| 206  | Wrought copper beryllium (WCUBE)                               |
| 207  | Cast copper alloys (CCUA)                                      |
| 300  | Titanium alloy (TA)  |
| 400  | Wrought magnesium alloys (WMGA)                                |
| 401  | Cast magnesium alloys (CMGA)                                   |
| 500  | Fusible alloys, solders (FUSSOL)                               |
| 600  | Cast zinc alloys (CZINCA)                                      |
| 700  | Wrought nickel alloys (WNIA)                                   |
| 701  | Cast nickel alloys (CNIA)                                      |
| 800  | Precious metals (PRECMET)                                      |

| CODE | Description                       |
|------|-----------------------------------|
| 900  | Clad materials (CLADMAT)          |
| 1000 | Thermoplastics (THERPLAS)         |
| 1001 | Thermosetting plastics (TSETPLAS) |

Table 9-23 Spot and Seam Weld Fatigue Analysis SN Curve Defaults

| Property   | Generic Nugget Spot Weld |                 | Generic Top/Bottom Sheet Spot Weld |                 | Generic Flexible Seam Weld (r=1) |                 | Generic Stiff Seam Weld (r=0) |                 |
|------------|--------------------------|-----------------|------------------------------------|-----------------|----------------------------------|-----------------|-------------------------------|-----------------|
| CODE       | steel<br><100            | alum<br>100-199 | steel<br><100                      | alum<br>100-199 | steel<br><100                    | alum<br>100-199 | steel<br><100                 | alum<br>100-199 |
| UTS (MPa)* | 4000                     | 4000            | 4000                               | 4000            | 1.8E4                            | 4000            | 1.8E4                         | 4000            |
| YS (MPa)   | 355                      | 150             | 355                                | 150             | Blank                            | Blank           | Blank                         | Blank           |
| E (MPa)    | 2.1E5                    | 7E4             | 2.1E5                              | 7E4             | 2.1E5                            | 7E4             | 2.1E5                         | 7E4             |
| SRI1 (MPa) | 2100                     | 2462            | 2900                               | 2462            | 3.6E4                            | 3140            | 1.8E4                         | 1275            |
| b1         | -0.1667                  | -0.2            | -0.1667                            | -0.2            | -0.3333                          | -0.1734         | -0.3333                       | -0.1625         |
| Nc1        | 1E6                      | 1E7             | 1E6                                | 1E7             | 1E7                              | 1E7             | 1E7                           | 1E7             |
| b2         | -0.09091                 | -0.1111         | -0.09091                           | -0.1111         | -0.3333                          | -0.1734         | -0.3333                       | -0.1625         |
| SE         | 0.334                    | 0.33            | 0.33                               | 0.33            | 0.4                              | 0.4             | 0.4                           | 0.4             |
| RR         | 0.0                      | 0.0             | 0.0                                | 0.0             | -1.0                             | -1.0            | -1.0                          | -1.0            |
| Nfc        | 1E30                     | 1E30            | 1E30                               | 1E30            | 1E30                             | 1E30            | 1E30                          | 1E30            |
| MSS        | 0.1                      | 0.2             | 0.1                                | 0.2             | Blank                            | Blank           | Blank                         | Blank           |
| M1         | Blank                    | Blank           | Blank                              | Blank           | Blank                            | Blank           | Blank                         | Blank           |
| M2         | -0.1                     | -0.2            | -0.1                               | -0.2            | -0.25                            | -0.25           | -0.25                         | -0.25           |
| M3         | -0.1                     | -0.2            | -0.1                               | -0.2            | -0.1                             | Blank           | -0.1                          | Blank           |
| M4         | Blank                    | Blank           | Blank                              | Blank           | -0.1                             | Blank           | -0.1                          | Blank           |
| BTHRESH    | Blank                    | Blank           | Blank                              | Blank           | Blank                            | Blank           | Blank                         | Blank           |
| RTHICK     | Blank                    | Blank           | Blank                              | Blank           | 1 (mm)                           | 1 (mm)          | 1 (mm)                        | 1 (mm)          |
| nTHICK     | Blank                    | Blank           | Blank                              | Blank           | 0.16667                          | 0.16667         | 0.16667                       | 0.16667         |

\*Both Spot Weld and Seam Weld fatigue analyses use structural stresses, which do not relate to UTS. They both should have static failure set to false or to ensure static failure is not active, the UTS should be set to a high value, thus the high default values for these data sets. Corrections based on UTS do not apply to welds.

Table 9-24 Steel Scale Factor, Diameter &amp; Thickness Exponent Defaults for Fatigue Analysis of Spot Welds

| Steel<br>(CODE < 100) | Scale Factor | Diameter Exponent | Thickness Exponent |
|-----------------------|--------------|-------------------|--------------------|
| FX,FY                 | SF-FXY = 1.0 | DE-FXY = 0.0      | TE-FXY = 0.0       |
| MX,MY                 | SF-MXY = 0.6 | DE-MXY = 0.0      | TE-MXY = 0.5       |
| FZ                    | SF-FZ = 0.6  | DE-FZ = 0.0       | TE-FZ = 0.5        |
| MZ                    | SF-MZ = 0.0  | DE-MZ = 0.0       | TE-MZ = 0.0        |

Aluminum Scale Factor, Diameter & Thickness Exponent Defaults  
for Fatigue Analysis of Spot Welds

| Aluminum<br>( $100 \leq \text{CODE} \leq 200$ ) | Scale Factor | Diameter Exponent | Thickness Exponent |
|---|--------------|-------------------|--------------------|
| FX,FY   | SF-FXY = 0.4 | DE-FXY = 0.5      | TE-FXY = -0.25     |
| MX,MY   | SF-MXY = 0.4 | DE-MXY = 0.5      | TE-MXY = -0.25     |
| FZ  | SF-FZ = 1.0  | DE-FZ = 0.0       | TE-FZ = 1.0        |
| MZ  | SF-MZ = 0.0  | DE-MZ = 0.0       | TE-MZ = 0.0        |

**MATG****Gasket Material Properties**

Specifies gasket material properties to be used in SOL 600 and SOL 400.

**Format:**

| 1    | 2      | 3      | 4       | 5      | 6      | 7      | 8      | 9      | 10     |
|------|--------|--------|---------|--------|--------|--------|--------|--------|--------|
| MATG | MID    | IDMEM  | BEHAV   | TABLD  | TABLUI | TABLUS | TABLUS | TABLUS | TABLUS |
|      | TABLUS | TABLUS | TABLUS  | TABLUS | TABLUS | TABLUS | YPRS   | EPL    |        |
|      | GPL    | GAP    | TABYPRS | TABEPL | TABGPL | TABGAP | N/A    | N/A    |        |

**Example:**

|      |      |     |   |      |      |      |      |       |  |
|------|------|-----|---|------|------|------|------|-------|--|
| MATG | 100  | 10  | 0 | 1001 | 1002 | 1003 |      |       |  |
|      |      |     |   |      |      |      | 100. | 2500. |  |
|      | 950. | 0.0 |   |      |      |      |      |       |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material ID number. (Integer)   |
| IDMEM      | ID of MAT1 providing material behavior for membrane behavior. See Remarks.<br>(Integer)   |
| BEHAV      | Behavior type (presently only type 0 is supported). (Integer)   |
| TABLD      | ID of a TABLES1 table providing loading path of the gasket (pressure versus displacement). See Remarks 1. and 3. (Integer)  |
| TABLUI     | ID of TABLES1 table providing unloading path(s) of the gasket (pressure versus displacement) can range from 1 to 10. If there is no unloading, no unloading tables need be entered. Leave fields blank for tables that are not required. See Remarks. (Integer) |
| YPRESS     | Yield pressure. See Remark 4. (Real)  |
| EPL        | Tensile modulus (pressure per unit length). (Real)  |
| GPL        | Transverse shear modulus (force per unit area). (Real)  |
| GAP        | Initial gap (if present). (Real)  |
| TABYPRS    | ID of TABLES1 table associated with yield pressure (not presently used). (Integer)  |
| TABEPL     | ID of TABLES1 table associated with tensile modulus (not presently used). (Integer)   |
| TABGPL     | ID of TABLES1 table associated with transverse shear modulus (not presently used). (Integer)  |
| TABGAP     | ID of TABLES1 table associated with initial gap (not presently used). (Integer)   |

**Remarks:**

1. MATG defines nonlinear properties in the thickness direction for compression only, designed for gasket-like materials. MATG has anisotropy only in the thickness direction, which is called normal anisotropy.
2. The MATG option can only be used with continuum composite elements, use PLCOMP.
3. The MATG entry defines the compressive behavior in thickness. The thickness direction is the principal direction (3) in 3-dimensional solids and (2) for 2-dimensional solids (plane strain and axisymmetric elements). Since MATG material allows only normal anisotropy, linear properties in MAT1 are required for in-plane behavior.
4. The initial yield pressure should match a point in table TABLD.
5. The loading path for the gasket is always in compression. However, it starts from the origin to initial yield pressure (nonlinear elastic range) and continues with strain hardening slope into the plastic region. All the data points are specified in the first quadrant.
6. As many as 10 unloading paths may be defined in the thickness direction using TABLS1 in pressure vs. gasket closure distance as in the loading path. All the unloading paths must start from zero pressure and end at the loading path in the plastic region. Unloading behavior at undefined paths will be interpolated between two adjacent unloading paths. The last point of the last specified unloading path signifies full compression, which does not allow any further closure beyond the point.

| Element Thickness Direction for MATG | Runs Parallel from Face | To Face     |
|--------------------------------------|-------------------------|-------------|
| Element T direction                  | G1-G2-G3-G4             | G5-G6-G7-G8 |

7. If creep analysis is required, Bulk Data entry, MPCREEP, must also be entered to activate Marc's CREEP parameter.

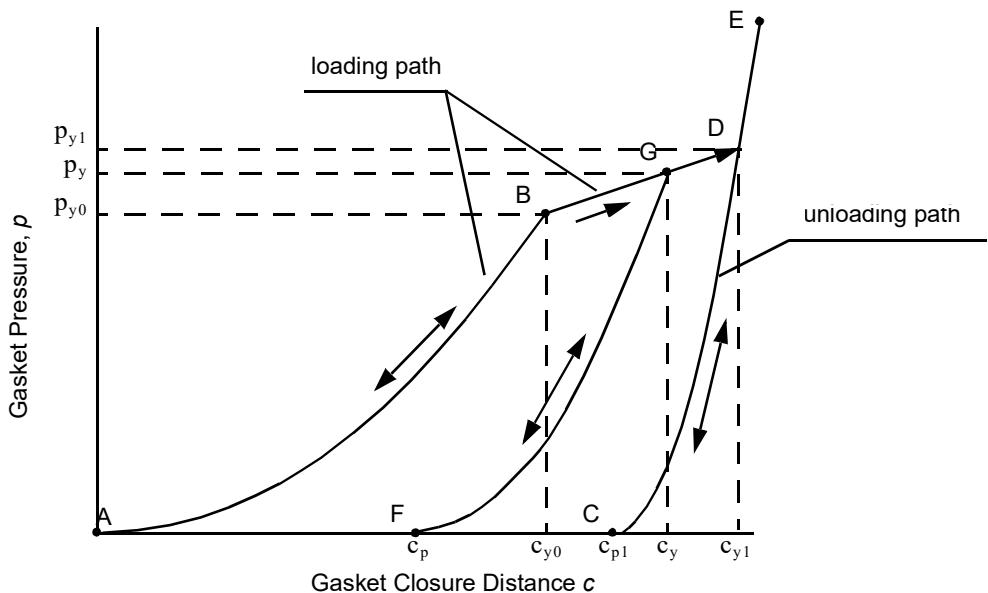


Figure 9-110 Pressure-closure Relation of a Gasket

8. See associated MATTG entry for temperature variation of these properties.
9. All continuation cards must be entered.
10. MID, IDMEM, BEHAV, TABLD, TABLU1, YPRS, EPL and GPL must be non-zero.
11. Each unloading curve must begin with gasket pressure of 0.0. Subsequent unloading curves must start with larger closure distances (when gasket pressure is 0.0) than previous unloading curves.
12. Points on loading and unloading curves must be defined in order of increasing gasket pressure.
13. MATG may be referenced by “solid” composite elements only, via PCOMPLS linked to PSOLID and PSHLN2 linked to PLPLANE.
14. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATTIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
15. Result data recovery is not supported in SOL 400 when ANALSYS=MODES, MFREQ or DFREQ.

**MATHE****Hyperelastic Material Properties**

Specifies hyperelastic (rubber-like) material properties for nonlinear (large strain and large rotation) analysis in SOL 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin Model (Model = Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

| 1     | 2   | 3     | 4   | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|-----|-------|-----|------|------|------|------|------|----|
| MATHE | MID | Model |     | K    | RHO  | Texp | Tref | GE   |    |
|       | C10 | C01   | D1  | TAB1 | TAB2 | TAB3 | TAB4 | TABD |    |
|       | C20 | C11   | C02 | D2   | NA   | ND   |      |      |    |
|       | C30 | C21   | C12 | C03  | D3   |      |      |      |    |
|       | C40 | C31   | C22 | C13  | C04  | D4   |      |      |    |
|       | C50 | C41   | C32 | C23  | C14  | C05  | D5   |      |    |

Format 2: (Model = Ogden or Foam)

| 1     | 2   | 3      | 4     | 5   | 6      | 7     | 8    | 9  | 10 |
|-------|-----|--------|-------|-----|--------|-------|------|----|----|
| MATHE | MID | Model  | NOT   | K   | RHO    | Texp  | Tref | GE |    |
|       | Mu1 | Alpha1 | Beta1 |     |        |       |      |    |    |
|       | Mu2 | Alpha2 | Beta2 | Mu3 | Alpha3 | Beta3 |      |    |    |
|       | Mu4 | Alpha4 | Beta4 | Mu5 | Alpha5 | Beta5 |      |    |    |
|       | D1  | D2     | D3    | D4  | D5     |       |      |    |    |

Note that Foam does not support D1~D5.

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

| 1     | 2   | 3     | 4  | 5  | 6   | 7    | 8    | 9  | 10 |
|-------|-----|-------|----|----|-----|------|------|----|----|
| MATHE | MID | Model |    | K  | RHO | Texp | Tref | GE |    |
|       | NKT | N/E   | Im |    |     |      |      |    |    |
|       | D1  | D2    | D3 | D4 | D5  |      |      |    |    |

Format 4: User Strain Energy Function (Model = GHDMI) (SOL 400 only)

| 1     | 2   | 3     | 4 | 5 | 6   | 7    | 8    | 9  | 10 |
|-------|-----|-------|---|---|-----|------|------|----|----|
| MATHE | MID | Model |   | K | RHO | Texp | Tref | GE |    |

## Example - Format 1:

|       |          |      |      |          |         |          |    |      |  |
|-------|----------|------|------|----------|---------|----------|----|------|--|
| MATHE | 2        | Foam |      | 3.30E+07 | 0.00035 | 4.00E-06 | 75 | 0.02 |  |
|       | 2.00E+04 | 1.5  | 1.15 |          |         |          |    |      |  |
|       | 1.50E+04 |      |      |          |         |          |    |      |  |

## Example - Format 2:

|       |    |       |      |        |  |  |  |  |  |
|-------|----|-------|------|--------|--|--|--|--|--|
| MATHE | 1  | Ogden | 2    | 200000 |  |  |  |  |  |
|       | 16 | 2     | 1.15 |        |  |  |  |  |  |
|       | -4 | -2    |      |        |  |  |  |  |  |

## Example - Format 3:

|       |          |      |  |          |         |          |    |      |  |
|-------|----------|------|--|----------|---------|----------|----|------|--|
| MATHE | 2        | Gent |  | 3.30E+07 | 0.00035 | 4.00E-06 | 75 | 0.02 |  |
|       | 2.00E+04 | 1250 |  |          |         |          |    |      |  |

## Example - Format 4:

|       |   |       |  |          |         |          |      |       |  |
|-------|---|-------|--|----------|---------|----------|------|-------|--|
| MATHE | 2 | GHEM1 |  | 3.30E+07 | 0.00035 | 2.50E-06 | 70.5 | 0.015 |  |
|-------|---|-------|--|----------|---------|----------|------|-------|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MATHE entry. (Integer > 0; no Default)  |
| Model      | Select hyperelastic material model from (Character; Default = Mooney):   |
| Mooney     | for generalized Mooney-Rivlin hyperelastic model. See Remark 1.  |
| Ogden      | for Ogden hyperelastic model. See Remark 2.  |
| Foam       | for foam model. See Remark 3.  |
| Aboyce     | for Arruda-Boyce strain energy model. See Remarks 4. and 17.   |
| Gent       | for Gent strain energy model. See Remarks 5. and 17.   |
| GHEM1      | Invariant-base user subroutine (foam models only)<br>$W = W(I_1, I_2, I_3)$  |
| GHEM2      | Principal-stretch based user subroutine (foam models only)<br>$W = W(\lambda_1, \lambda_2, \lambda_3)$   |
| GHEM3      | Invariant-based with volumetric and deviatoric split user subroutine (foam models only)<br>$W = W_{dev}(I_1, I_2) + U(J)$  |
| GHEM4      | Principal-stretch based with volumetric and deviatoric split user subroutine (foam models only)<br>$W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$ |
| GHEM5      | Invariant-based with deviatoric part only (rubber models only)<br>$W = W_{dev}(I_1, I_2)$  |

| Descriptor | Meaning  |   |   |   |   |   |   |   |   |   |   |   |  |
|------------|--|---|---|---|---|---|---|---|---|---|---|---|--|
| GHEM6      | <p>Principal-stretch based with deviatoric part only (rubber models only)</p> $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$ <p>where <math>I_1, I_2, I_3</math> (<math>\lambda_1, \lambda_2</math>, and <math>\lambda_3</math>) are strain invariants (principal stretches), and <math>\bar{I}_1</math> and <math>\bar{I}_2</math> (<math>\bar{\lambda}_1, \bar{\lambda}_2</math>, and <math>\bar{\lambda}_3</math>) are their deviatoric parts, defined by and <math>\bar{I}_2 = J^{-4/3}I_2</math> (<math>\bar{\lambda}_i = J^{-1/3}\lambda_i</math>, <math>i = 1, 2, 3</math>); <math>J</math> is the determinant of the deformation gradient.</p> <p>For GHEMi models, see Remarks 13., 14., and 15.</p>  |   |   |   |   |   |   |   |   |   |   |   |  |
| NOT        | Number of terms to be used in Ogden and Foam model. ( $0 < \text{Integer} \leq 5$ ; Default = 1)   |   |   |   |   |   |   |   |   |   |   |   |  |
| K          | Specifies a bulk modulus. See Remarks 7. and 16. (Real > 0 or -1.0; Default = automatically set for nearly incompressible condition. Enter -1.0 if nonlinear volumetric strain energy function defined by Di will be used.)  |   |   |   |   |   |   |   |   |   |   |   |  |
| RHO        | Mass density in original configuration. (Real; Default = 0.0)  |   |   |   |   |   |   |   |   |   |   |   |  |
| Texp       | Coefficient of thermal expansion. See Remark 8. (Real; Default = 0.0)  |   |   |   |   |   |   |   |   |   |   |   |  |
| Tref       | Reference temperature at which the thermal expansion coefficient is measured. Tref is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)   |   |   |   |   |   |   |   |   |   |   |   |  |
| GE         | <p>If GE is real it represents the structural damping coefficient. See Remark 12. (Real; Default = 0.0).</p> <p>For SOL 600 only, if GE is integer, it represents the form of the user subroutine uelastomer.f as follows:</p> <table> <tbody> <tr> <td>1</td><td>Foam Material - Invariant-based model<br/><math>W = W(I_1, I_2, I_3)</math></td></tr> <tr> <td>2</td><td>Foam Material - Principal stretch based<br/><math>W = W(\lambda_1, \lambda_2, \lambda_3)</math></td></tr> <tr> <td>3</td><td>Foam Material -Invariant-based with volumetric and deviatoric split<br/><math>W = W_{dev}(I_1, I_2) + U(J)</math></td></tr> <tr> <td>4</td><td>Foam Material - Principal stretch with volumetric and deviatoric split<br/><math>W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)</math></td></tr> <tr> <td>5</td><td>Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based<br/><math>W = W_{dev}(\bar{I}_1, \bar{I}_2)</math></td></tr> <tr> <td>6</td><td>Rubber Material (Ogden) - Principal stretch based)<br/><math>W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)</math></td></tr> </tbody> </table> <p>where <math>I_1, I_2</math>, and <math>I_3</math> (<math>\lambda_1, \lambda_2</math>, and <math>\lambda_3</math>) are strain invariants (principal stretches), and <math>\bar{I}_1</math> and <math>\bar{I}_2</math> (<math>\bar{\lambda}_1, \bar{\lambda}_2</math>, and <math>\bar{\lambda}_3</math>) are their deviatoric parts, defined by <math>\bar{I}_1 = J^{-2/3}I_1</math> and <math>\bar{I}_2 = J^{-4/3}I_2</math> (<math>\bar{\lambda}_i = J^{-1/3}\lambda_i</math>, <math>i = 1, 2, 3</math>); <math>J</math> is the determinant of the deformation gradient.</p> <p>If GE is integer, Bulk Data entry USRSUB6 is also required (see Marc Vol D for more details.)</p> | 1 | Foam Material - Invariant-based model<br>$W = W(I_1, I_2, I_3)$ | 2 | Foam Material - Principal stretch based<br>$W = W(\lambda_1, \lambda_2, \lambda_3)$ | 3 | Foam Material -Invariant-based with volumetric and deviatoric split<br>$W = W_{dev}(I_1, I_2) + U(J)$ | 4 | Foam Material - Principal stretch with volumetric and deviatoric split<br>$W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$ | 5 | Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based<br>$W = W_{dev}(\bar{I}_1, \bar{I}_2)$ | 6 | Rubber Material (Ogden) - Principal stretch based)<br>$W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$ |
| 1          | Foam Material - Invariant-based model<br>$W = W(I_1, I_2, I_3)$  |   |   |   |   |   |   |   |   |   |   |   |  |
| 2          | Foam Material - Principal stretch based<br>$W = W(\lambda_1, \lambda_2, \lambda_3)$  |   |   |   |   |   |   |   |   |   |   |   |  |
| 3          | Foam Material -Invariant-based with volumetric and deviatoric split<br>$W = W_{dev}(I_1, I_2) + U(J)$  |   |   |   |   |   |   |   |   |   |   |   |  |
| 4          | Foam Material - Principal stretch with volumetric and deviatoric split<br>$W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$  |   |   |   |   |   |   |   |   |   |   |   |  |
| 5          | Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based<br>$W = W_{dev}(\bar{I}_1, \bar{I}_2)$  |   |   |   |   |   |   |   |   |   |   |   |  |
| 6          | Rubber Material (Ogden) - Principal stretch based)<br>$W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)$   |   |   |   |   |   |   |   |   |   |   |   |  |

| Descriptor | Meaning  |
|------------|--|
| NA         | Order of the distortional strain energy polynomial function. See Remark 6. (0 < Integer < or =5; Default = 2; Default = 5 for SOL 600)   |
| ND         | Order of the Volumetric strain energy polynomial function. See Remark 6. (0 < Integer < or =5; Default = 1; Default = 5 for SOL 600)   |
| Cij        | Material constants related to distortional deformation for generalized Mooney-Rivlin model. (Real; Default = 0.0) SOL 600 uses only five constants (C10, C01, C20, C11, and C30) and ignores others.   |
| Di         | Material constants related to volumetric deformation. (Real $\geq$ 0.0) Needed only if K = -1.0. Not used for foam model. See Remark 7. For SOL 600, only Mooney model supports nonlinear volumetric strain energy function.   |
| TAB1       | Table identification number of a TABLES1 entry that contains simple tension-compression data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios $l/l_0$ and y-values must be values of the engineering stress $F/A_0$ . $l_0$ is the initial length and $A_0$ is the initial cross-sectional area. See Remark 9. (Integer > 0 or blank)                          |
| TAB2       | Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios $l/l_0$ and y-values must be values of the engineering stress $F/A_0 \cdot l_0$ . $l_0$ is the initial length and $A_0$ is the initial cross-sectional area. See Remark 9. (Integer > 0 or blank)                       |
| TAB3       | Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be values of the shear strain and y-values must be values of the engineering shear stress. (Integer > 0 or blank)  |
| TAB4       | Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Cij. The x and y values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and the values of the nominal stress $F/A_0 \cdot l_0$ and $A_0$ are the initial length and cross-sectional area, respectively, in the l-direction. See Remark 9. (Integer > 0 or blank)               |
| TABD       | Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constant Di. The x-values in the TABLES1 entry must be values of the volume ration $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; y-values must be values of the pressure, assumed positive in compression. See Remark 9. (Integer > 0 or blank) |
| Muk        | Coefficients $\mu_k$ of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)   |
| Alphak     | Coefficients $\alpha_k$ of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)  |
| Betak      | Coefficients $\beta_k$ of the strain energy function for foam material. These fields should be left blank for Ogden mode. See Remarks 2. and 3. (Real; Default = 0)  |

| Descriptor | Meaning  |
|------------|--|
| NTK        | Material constant for Arruda-Boyce strain energy model. (Real > 0; Default = 1)  |
| N/E        | Material constant representing the number (N) of statistical links of the chain for Arruda-Boyce model; or tensile modulus (E) for Gent strain energy model. (Real > 0; Default = 1) |
| Im         | Maximum first invariant for Gent strain energy model. (Real > 0; Default = 0)  |

**Remarks:**

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$W(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^3 C_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + 4.5K(J^{1/3} - 1)^2$$

with

$$\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$$

$$\bar{I}_2 = \bar{\lambda}_1^2 \bar{\lambda}_2^2 + \bar{\lambda}_2^2 \bar{\lambda}_3^2 + \bar{\lambda}_3^2 \bar{\lambda}_1^2$$

where K and J are bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is related to the Mooney-Rivlin constants by

$$G = 2(C_{10} + C_{01}) \text{ with } C_{01} \approx 0.25 C_{10}$$

The model reduces to a Mooney-Rivlin material with only two constants (C10 and C01), and to a Neo-Hookean material with one constant (C10). The third order Mooney-Rivlin model in SOL 600 uses only five distortional constants (C10, C01, C11, C20, C30) and the bulk modulus K for volumetric deformation. Instead of MATHE, the hyperelastic material can be specified using MATHP Bulk Data entry in SOLs 106, 129, and 600.

2. For the Ogden material model, the strain energy function is

$$W = \sum_{k=1}^5 \frac{\mu_k}{\alpha_k} (\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3) + 4.5K(J^{1/3} - 1)^2$$

where  $\alpha_k$  and  $\mu_k$  are the moduli and exponent constants, while  $\bar{\lambda}_i$  is the deviatoric stretch ratio defined as:

$$\bar{\lambda}_i = J^{-\frac{1}{3}} \lambda_i$$

and J and K are the determinant of the deformation gradient and the bulk modulus, respectively. A two-term Ogden model is equivalent to a simple Mooney-Rivlin model

$$\mu_1 = 2C_{10} \quad \text{and} \quad \mu_2 = 2C_{01}$$

with       $\alpha_1 = 2$ .      and       $\alpha_2 = 2$ .

3. For foam material model, the distortional strain energy function is the same as Ogden. The volumetric energy function of foam model is defined by a polynomial function with coefficients  $\beta_i$
4. For the Arruda-Boyce model, the strain energy function is

$$W = N_{KT} \left[ \frac{1}{2}(\bar{I}_1 - 3) + \frac{1}{20N}(\bar{I}_1^2 - 9) + \frac{1}{1050N^2}(\bar{I}_1^3 - 27) \right. \\ \left. + \frac{19}{7000N}(\bar{I}_1^4 - 81) + \frac{519}{67375N^4}(\bar{I}_1^5 - 243) \right] + 4.5K(J^{1/3} - 1)^2$$

with  $\bar{I}_1 = \tilde{\lambda}_1^2 + \tilde{\lambda}_2^2 + \tilde{\lambda}_3^2$

where  $N_{KT}$  is a material constant and  $N$  is a material parameter representing the number of statistical links of the material chain.

If the material test data are available from multiple experiments such as uniaxial and equi-biaxial tests, the Ogden model is more accurate in fitting experimental results. If only uniaxial tension data is available, the Arruda-Boyce model provides more accurate data fitting for multiple modes of deformation.

5. For the Gent model, the strain energy function is

$$W = -\frac{1}{6}EI_m \log \left[ \frac{I_m}{I_m - \bar{I}_1 + 3} \right]$$

where  $E$  and  $I_m$  are tensile modulus and maximum first invariant, respectively.

6. The curve fitting is activated if any of the TAB1, TAB2, TAB3, TAB4 and/or TABD are specified. The NA and ND fields are used in case of the curve fitting for Mooney from the experimental data.
7. Although the conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible, SOL 600 provides a compressible rubber model. Nearly incompressible material may be simulated with a large value of  $K$ . The default value for the Mooney-Rivlin model represents a nearly incompressible condition, which is  $K = 10^4(C_{10} + C_{01})$ . In case of Ogden model, the default is  $2500 \cdot \sum \mu_k \alpha_k$ . If any of the  $D_i$  are non-zero, then the entered value of the Bulk Modulus must be -1.0 and the nonlinear volumetric strain energy function will be used. See Remark 16.
8. The thermal expansion coefficient is a secant value measured with respect to a temperature, Tref.

The thermal strain is computed by  $\varepsilon_{th} = \bar{\alpha}(T - T_0)$

where  $T_0$  is an initial temperature. The secant coefficient of thermal expansion is related to the instantaneous coefficient of thermal expansion by

$$\alpha = \frac{d\varepsilon_{th}}{dT} = \bar{\alpha} + \frac{d\bar{\alpha}}{dT}(T - T_0)$$

The choice of secant or instantaneous coefficient (CTE) of thermal expansion is determined by PARAM,MARALPHA in SOL 600. All CTE's in SOL 600 models must be either secant or instantaneous.

9. All the material constants may be obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties. The bulk modulus K may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 fields are blank, the material constants must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters. It is recommended that the GUI be used to determine the materials constants, so one can visualize the correlation between the numerical representation and the experimental data. It is recommended that the experimental data span the range of anticipated strains to reduce the chances of adverse numerical calculations.
10. All the alphanumeric fields are recognizable by the first four letters.
11. Enter NKT and N/E for Aboyce (leave Im blank). Enter N/E and Im for Gent (leave NKT blank).
12. For SOL 600, Bulk Data entry USRSUB6 is also required.
13. The GHEMi models using the user subroutine are only available if the updated Lagrange formulation is used for the hyperelastic material. The user subroutine MD UELASTOMER defined with the GHEMi must be used along with the Bulk Data entry MATUDS.
14. The GHEMi are not available for plane stress, shell, membrane, and beam elements.
15. For the GHEM3 and GHEM4 models, the user subroutine is called twice, once with either a 3 or 4 to evaluate the deviatoric strain energy and once with a 7 to evaluate the volumetric strain energy.
16. The series representation of the volumetric strain energy is only available for the updated Lagrange formulation.
17. The Gent and Arruda-Boyce models are not available for plane stress, shell, membrane, and beam elements.
18. The Ogden, Foam, Arruda-Boyce and Gent models are not available in SOLs 106 and 129.
19. In SOL 400, MATHE is only supported for nonlinear elements with property extensions. This implies that for such elements, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for incompressible rubber models, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral. Note that only PSOLID with PSLDN1 should be used for 3D solid nonlinear elements and PLSOLID cannot be used for MATHE. Note also that,

prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

20. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**Notes (SOL 600 only):**

1. The structural damping constant GE is specified in Marc under the option DAMPING as a numerical damping  $\gamma$ , i.e.,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE;  $\alpha$  and  $\beta$  are equivalent to user parameters ALPHA1 and ALPHA2.

2. Material curve-fitting from experimental data is not supported in SOL 400. Direct material property input is required.
3. This Bulk Data entry accommodates Marc's input data under the model definition options MOONEY, OGDEN, and FOAM as well as the parameter ELASTICITY. It also accommodates MATHP input data in Nastran.

**MATHE****Hyperelastic Material Properties - SOL 700**

Specifies hyperelastic (rubber-like) material properties for nonlinear analysis. Use SOL700 only.

**Format 1 (default): Mooney-Rivlin Model (Model=Mooney1)**

| 1     | 2   | 3     | 4  | 5 | 6   | 7 | 8 | 9 | 10 |
|-------|-----|-------|----|---|-----|---|---|---|----|
| MATHE | MID | Model |    |   | RHO |   |   |   |    |
|       | A   | B     | NU |   |     |   |   |   |    |

**Format 2: Ogden Model (Model=Ogden1)**

| 1     | 2   | 3      | 4  | 5   | 6      | 7 | 8 | 9 | 10 |
|-------|-----|--------|----|-----|--------|---|---|---|----|
| MATHE | MID | MODEL  |    |     | RHO    |   |   |   |    |
|       | MU1 | ALPHA1 | NU |     |        |   |   |   |    |
|       | MU2 | ALPHA2 |    | MU3 | ALPHA3 |   |   |   |    |
|       | MU4 | ALPHA4 |    | MU5 | ALPHA5 |   |   |   |    |
|       | MU6 | ALPHA6 |    | MU7 | ALPHA7 |   |   |   |    |
|       | MU8 | ALPHA8 |    |     |        |   |   |   |    |

**Format 3: Foam Model (Model=FOAM1)**

| 1     | 2     | 3      | 4      | 5     | 6    | 7 | 8 | 9 | 10 |
|-------|-------|--------|--------|-------|------|---|---|---|----|
| MATHE | MID   | Model  |        | K     | RHO  |   |   |   |    |
|       | TYPE  | VALUE  | CUTOFF | TABLE | TABY |   |   |   |    |
|       | ALPHA | UNLOAD |        |       |      |   |   |   |    |

**Example –Format 1:**

|       |      |         |       |  |      |  |  |  |  |
|-------|------|---------|-------|--|------|--|--|--|--|
| MATHE | 7    | MOONEY1 |       |  | 100. |  |  |  |  |
|       | 0.34 | 0.27    | 0.495 |  |      |  |  |  |  |

**Example –Format 2:**

|       |      |        |       |      |      |  |  |  |  |
|-------|------|--------|-------|------|------|--|--|--|--|
| MATHE | 7    | OGDEN1 |       |      | 100. |  |  |  |  |
|       | 6.3  | 1.3    | 0.495 |      |      |  |  |  |  |
|       | 0.12 | 5.0    |       | -0.1 | -2.0 |  |  |  |  |

**Example –Format 3:**

|       |     |        |     |     |      |  |  |  |  |
|-------|-----|--------|-----|-----|------|--|--|--|--|
| MATHE | 7   | FOAM1  |     | 3.  | 100. |  |  |  |  |
|       | 1.0 | -100.0 | 1.0 | 111 | 112  |  |  |  |  |
|       | 0.4 | 2.0    |     |     |      |  |  |  |  |

| Descriptor    | Contents  |
|---------------|---|
| MID           | Material identification number. (Integer > 0; required.)  |
| Model         | Select hyperelastic material model from (Character; Default = Mooney1)  |
| Mooney1       | Mooney-Rivlin hyperelastic model. It is only available for solid elements. See Remark 1.  |
| Ogden1        | Ogden hyperelastic model. It is only available for solid elements. See Remark 2.  |
| FOAM1         | Isotropic elastic foam material. It is only available for solid elements. See Remarks 3.-8.   |
| RHO           | Density. (Real > 0; required)   |
| A             | Strain-energy density function constant. (Real; required)   |
| B             | Strain-energy density function constant. (Real; required)   |
| NU            | Poisson's ratio. ( $0.0 \leq R < 0.5$ ; required)   |
| MU1-MU8       | Strain energy density function coefficients. MU1 is required. (Real; default=0.0)   |
| ALPHA1-ALPHA8 | Strain energy density function coefficients. ALPHA1 is required. (Real; default=0.0)  |
| TYPE          | The type of data defined as the x-value in the table. (Real > 0.0; default=1.0) <ul style="list-style-type: none"> <li>1.0      Crush factor. (= 1-relative volume)</li> <li>2.0      True strain</li> </ul>                                  |
| VALUE         | The value for cut-off stress. (Real≤0.0; default=-0.1×Young's modulus)  |
| CUTOFF        | Cut-off stress. (Real > 0.0; default=3.0) <ul style="list-style-type: none"> <li>1.0      Stress for tensile failure</li> <li>2.0      Minimum stress</li> <li>3.0      Not used</li> </ul>   |
| TABLE         | TABLED1 ID defining the variation of stress (y-value) with crush factor or true strain (x- value). (Integer > 0; required)  |
| TABY          | TABLED1 ID giving the variation of the scale factor for the stress (y-value) with the strain rate (x-value). (Integer > 0; default=not used)  |
| ALPHA         | Energy dissipation factor. ( $0.0 \leq \text{Real} \leq 1.0$ ; default=0.0)   |
| UNLOAD        | Unloading option. (Real > 0.0; default=3.0) <ul style="list-style-type: none"> <li>1.0      Unloading via exponential curve</li> <li>2.0      Unloading via piecewise linear curve</li> <li>3.0      Unloading via quadratic curve</li> </ul> |

**Remarks:**

1. The Mooney-Rivlin constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components rather than by Hooke's law.

The strain-energy density function is formulated according to the Mooney-Rivlin model and is defined as

$$W(I_1, I_2, I_3) = A(I_1 - 3) + B(I_2 - 3) + C\left(\frac{1}{I_3^2} - 1\right) + D(I_3 - 1)^2$$

The constants C and D are defined as:

$$\begin{aligned} C &= \frac{1}{2}A + B \\ D &= \frac{A(5v - 2) + B(11v - 5)}{2(1 - 2v)} \end{aligned}$$

where A, B and v are input parameters.

$I_1$ ,  $I_2$  and  $I_3$  are strain invariants in terms of stretches.

For a rubber-like material, the shear modulus G is much less than the bulk modulus K. As a result, Poisson's ratio is nearly equal to one half.

2. The constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components using Ogden function.

The strain-energy density function is formulated according to the Ogden model and is defined as

$$W = \sum_{i=1}^3 \sum_{j=1}^8 \frac{\mu_j}{\alpha_j} (\bar{\lambda}_i^{\alpha_j} - 1) + \frac{1}{2}K(J-1)^2$$

The over bar (-) indicates that the volumetric effects have been eliminated from the principal stretches.

3. Poisson's ratio for FOAM1 model is effectively zero. Therefore, the shear and elastic moduli are calculated from K, the bulk modulus.
4. For this model, the stress-strain curve is independent of the experimental test performed to obtain the material data (uniaxial, shear, or volumetric). The most common test is the uniaxial compression test where the stress-strain characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain, or in terms of the true strain. Since Poisson's ratio is effectively zero, the amount of crush is defined as  $\left(1 - \frac{V}{V_0}\right)$ , with  $\left(\frac{V}{V_0}\right)$  as the relative volume, and the true strain is defined as.

$$\int_{t_0}^t \frac{dV}{V}$$

or

$$\ln\left(\frac{V}{V_0}\right)$$

The crush factor must be between 0 and 1. The true strain must always be negative and the stress positive (absolute value).

5. The yield surface in three-dimensional space is a sphere in principal stresses, and is defined by

$$\tau_{11}^2 + \tau_{22}^2 + \tau_{33}^2 = R_s^2$$

where the radius of the sphere,  $R_s$ , depends on the strains and strain rates as follows:

$$R_s = f_1(R_e)f_2(R_r)$$

with

$$R_e = \varepsilon_{11}^2 + \varepsilon_{22}^2 + \varepsilon_{33}^2$$

and

$$R_r = \dot{\varepsilon}_{11}^2 + \dot{\varepsilon}_{22}^2 + \dot{\varepsilon}_{33}^2$$

and  $f_1$  is the function supplied in the stress-strain table and  $f_2$  (if defined) is the function supplied in the factor-strain rate table.

6. A minimum (CUTOFF=2.0) or failure (CUTOFF=1.0) tensile stress can be defined. In the first case this corresponds to a tensile cut-off where the stress cannot fall below the minimum value. In the second case, if the stress falls below the failure stress the element fails and cannot carry tensile loading for the remainder of the analysis. Thus the stress can never become negative again.
7. The unloading behavior is piecewise linear (UNLOAD=2.0), quadratic (UNLOAD=3.0) or exponential (UNLOAD=1.0). The unloading curve is constructed such that the ratio of the dissipated energy (area between compressive loading and unloading curve) to total energy (area under the loading curve) is equal to the energy dissipation factor alpha. In the case of piecewise linear unloading, Nastran SOL700 constructs an unloading curve whose segments are parallel to the supplied compression table, except for the first and last segments, which pass respectively through the origin and the point P on the compression curve where the unloading starts. In the case of quadratic unloading, Nastran SOL700 constructs a quadratic curve starting in the origin and ending in point P. If the quadratic unloading curve falls below the strain axis, then the unloading stress is set to zero. In the case of exponential unloading, the unloading curve is constructed in a similarly to quadratic unloading except for the shape of the curve, which is created from an exponential function instead of a quadratic polynomial.
8. Not default values for taby, alpha, unload options are allowed only when cutoff is set to 1.0 or 2.0.

**MATHED**

## Damage Model Properties for Hyperelastic Materials - SOL 600

Specifies damage model properties for hyperelastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

Format:

| 1      | 2    | 3      | 4       | 5       | 6     | 7       | 8       | 9     | 10 |
|--------|------|--------|---------|---------|-------|---------|---------|-------|----|
| MATHED | MID  | method | Scale1  | Relax1  | Prop1 | Scale2  | Relax2  | Prop2 |    |
|        | Dinf | N/A    | Scalem1 | Relaxm1 | N/A   | Scalem2 | Relaxm2 |       |    |

Example:

|        |     |  |    |     |     |     |     |     |  |
|--------|-----|--|----|-----|-----|-----|-----|-----|--|
| MATHED | 100 |  | 1. | 0.3 | 0.5 | 0.8 | 0.4 | 1.0 |  |
|--------|-----|--|----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of MATHE entry. See Remark 1. (Integer > 0)  |
| Method     | Select a method for damage calculation (Character) from:<br>Multip for multiplicative decomposition. (Default)<br>Additiv for additive decomposition. See Remarks 2. and 3.  |
| Scale1,2   | Scaling factor $d_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0.0$ ; < 1.0; Default = 0.0)  |
| Relax1,2   | Relaxation rate $\eta_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0$ or blank)  |
| Prop1,2    | Proportionality factor $\delta_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. See Remark 4. (Real > 0.0; Default = 1.0)  |
| Dinf       | $d^\infty$ described in the equations of Remark 3. If Blank, the program will compute it; however even if set, the program usually calculates it. In most cases $d^\infty = 1.0 - scale1 - scale2$ . (Real > 0.0 or blank) |
| Scalem1,2  | Scaling factor $d_m$ for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0.0$ ; Default = 0.0)   |
| Relaxm1,2  | Relaxation rate $\lambda_m$ for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real > 0.0 or blank)  |

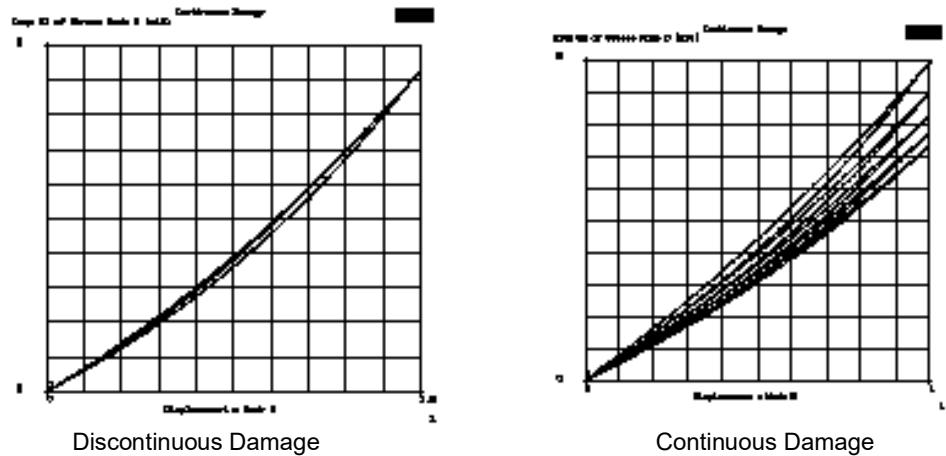
Remarks:

1. The MATHE Bulk Data entry with the same MID must exist for MATHED to be effective.  
 The damage capability is available for all the elastomeric materials (Mooney-Rivlin, Ogden, Gent, Arruda-Boyce).

2. Under repeated application of loads, elastomers undergo damage by mechanisms involving chain breakage, multi-chain damage, micro-void formation, and micro-structural degradation due to detachment of filler particles from the network entanglement. The damage model for elastomeric materials is based on the undamaged strain energy function  $W_0$ , multiplied by a Kachanov damage factor,  $K$ , i.e.,

$$W = K(\alpha, \beta)W_0$$

where  $\alpha$  and  $\beta$  are parameters for discontinuous and continuous damage models, respectively. Two types of phenomenological models, discontinuous and continuous, exist to simulate the damage. The discontinuous damage model simulates the "Mullins' effect," which involves a loss of stiffness (represented by a parameter  $\alpha$ ) below the previously attained maximum strain. The higher the maximum attained strain the larger the loss of stiffness is. There is a progressive stiffness loss with increasing maximum strain amplitude. Most of the stiffness loss takes place in the first few cycles provided the maximum strain level is not increased. This phenomenon is observed in both filled as well as natural rubber although the higher level of carbon black particles increases the hysteresis and the loss of stiffness. The continuous damage model (Miehe's formulation) can simulate the damage accumulation for strain cycles for which the values of effective energy is below the maximum attained value of the past history. The evolution of continuous damage parameter is governed by the arc-length of the effective strain energy, represented by a parameter  $\beta$ .



3. Both the continuous damage as well as the discontinuous damage can be modeled by a cumulative Kachanov factor in multiplicative or additive decomposition form.

For multiplicative decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right)$$

For additive decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n^\alpha \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_{m=1}^2 d_m^\beta \exp\left(-\frac{\beta}{\lambda_m}\right)$$

where  $d_n$ ,  $\delta_n$ ,  $\eta_n$ ,  $d_m$ , and  $\lambda_m$  are constants specified by the user, and  $d^\infty$  is calculated by the program such that the Kachanov factor assumes a value of unity at zero damage if left blank.

4. The proportionality factor is not used by additive decomposition which requires the continuation fields to include the continuous damage model.
5. User subroutines must be called out using PARAM,MARCUSUB,CHAR where CHAR is a character variable such as UDAMAG.

**Note:**

1. This Bulk Data entry accommodates Marc's input data under the model definition option DAMAGE.

**MATHP****Hyperelastic Material Properties**

Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).

**Format:**

| 1     | 2    | 3    | 4    | 5    | 6   | 7   | 8    | 9    | 10 |
|-------|------|------|------|------|-----|-----|------|------|----|
| MATHP | MID  | A10  | A01  | D1   | RHO | AV  | TREF | GE   |    |
|       |      | NA   | ND   |      |     |     |      |      |    |
|       | A20  | A11  | A02  | D2   |     |     |      |      |    |
|       | A30  | A21  | A12  | A03  | D3  |     |      |      |    |
|       | A40  | A31  | A22  | A13  | A04 | D4  |      |      |    |
|       | A50  | A41  | A32  | A23  | A14 | A05 | D5   |      |    |
|       | TAB1 | TAB2 | TAB3 | TAB4 |     |     |      | TABD |    |

**Example:**

|       |   |     |    |  |  |  |  |  |  |
|-------|---|-----|----|--|--|--|--|--|--|
| MATHP | 2 | 80. | 20 |  |  |  |  |  |  |
|       |   | 1   | 1  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MATHP entry. (Integer > 0; No Default)  |
| Aij        | Material constants related to distortional deformation. (Real; Default = 0.0)  |
| Di         | Material constants related to volumetric deformation. (Real $\geq 0$ ; Default for D1 is $10^3 \cdot (A10 + A01)$ ; Default for D2 through D5 is 0.0.)   |
| RHO        | Mass density in original configuration. (Real; Default = 0.0)  |
| AV         | Coefficient of volumetric thermal expansion. (Real; Default = 0.0)   |
| TREF       | Reference temperature. See <a href="#">MATBV, 2206</a> . (Real; Default = 0.0)   |
| GE         | Structural damping element coefficient. (Real; Default = 0.0)  |
| NA         | Order of the distortional strain energy polynomial function. ( $0 < \text{Integer} \leq 5$ ; Default = 1)  |
| ND         | Order of the volumetric strain energy polynomial function. ( $0 < \text{Integer} \leq 5$ ; Default = 1)  |
| TAB1       | Table identification number of a TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios $l/l_0$ and yi values must be values of the engineering stress $F/A_0$ . Stresses are negative for compression and positive for tension. (Integer > 0 or blank) |

| Descriptor | Meaning   |
|------------|---|
| TAB2       | Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios $l/l_0$ . yi values must be values of the engineering stress $F/A_0$ . l is the current length, F is the current force, $l_0$ is the initial length and $A_0$ is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $P r_0 \lambda^2 / 2 t_0$ where P = current value of the pressure and $r_0$ , $t_0$ = initial radius and thickness. (Integer > 0 or blank) |
| TAB3       | Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent $\gamma$ and yi values must be values of the engineering shear stress $F/A_0$ . (Integer > 0 or blank)  |
| TAB4       | Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Aij. xi and yi values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and values of the nominal stress $F/A_0$ . l is the current length, F is the current force, $l_0$ and $A_0$ are the initial length and cross-sectional area, respectively in the 1-direction. (Integer > 0 or blank)  |
| TABD       | Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression. (Integer > 0 or blank)  |

### Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{NA} A_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \sum_{i=1}^{ND} D_i (J - 1 - AV(T - T_0))^{2i}$$

$$A00 = 0$$

where  $\bar{I}_1$  and  $\bar{I}_2$  are the first and second distortional strain invariants, respectively;  $J = \det F$  is the determinant of the deformation gradient; For small strains, the bulk modulus K and the shear modulus G are related to the Mooney-Rivlin Material.

$2D1 = K$  and  $2(A10 + A01) = G$  at small strains, in which  $K$  is the bulk modulus and  $G$  is the shear modulus. The model reduces to a Mooney-Rivlin material if  $NA=1$  and to a Neo-Hookean material if  $NA = 1$  and  $A01 = 0.0$ . See Remark 2. For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required. T is the current temperature and  $T_0$  is the initial temperature.

2. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not presently available but may be simulated with a large enough value of D1. Note that a value of D1 lower than  $10^3 \cdot (A10 + A01)$  is, however, not recommended.
3. The material constants  $A_{ij}$  and  $D_i$  are obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties  $A_{ij}$ .  $D_i$  may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank,  $A_{ij}$  must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supercedes the manual input of the parameters.
4. IF ND=1 and a nonzero value of D1 is provided or obtained from experimental data in TABD, then the parameter estimation of the material constants  $A_{ij}$  takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimating the material constants.
5. See Chapters [Hyperelastic Elements](#), 197, [Hyperelastic Material](#), 295 and [Hyperelastic Material](#) in the *MSC Nastran Reference Guide* for further details.
6. In SOL400, MATHP is supported for 2 sets of elements:
  - a. Incompressible conventional 2D (plane strain and axisymmetric) elements and 3D (continuum) elements without property extensions. They are prescribed through PLPLANE and PLSOLID, respectively. The original element formulations that are also available in SOL106 are used in these cases.
  - b. Incompressible elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively.  
Also, for incompressible rubber models, 2D 3-node triangular elements and 3D 4-node tetrahedral elements should be associated with an incompressible formulation by specifying IPS for 2D plane strain and IAX for 2D axisymmetric elements in PSHLN2, as well ISOL for 3D tetrahedral element in PSLDN1.
7. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**Note:**

1. PLSOLID is only used with the elements in (a) and only PSOLID with PSLDN1 can be used for 3D solid nonlinear elements in (b).
2. Prior to MD Nastran 2010, if the property extensions were missing, the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATNLE****Advanced NLELAST Options - SOL 600**

The MATNLEX entries specify advanced forms of nonlinear elastic materials. The simple NLELAST option uses MAT1, MATS1, TALBES1 in the same manner as data is entered for SOL 106 or SOL 129.

**Format 1: Standard Nastran NLELAST Model**

Use NLELAST on MATS1 entry.

**Format 2: Invariant-based Material Model**

| 1       | 2    | 3    | 4   | 5    | 6     | 7  | 8     | 9     | 10 |
|---------|------|------|-----|------|-------|----|-------|-------|----|
| MATNLE2 | MID  | E    | PR  | RHO  | ALPH  | G  | TLIM  | CLIM  |    |
|         |      | TE   | TPR | TRHO | TALPH | TG | TTLIM | TCLIM |    |
|         | KIND | TYPE | LIM | GE   | NAME  |    |       |       |    |

**Format 3: Principal Strain-Based Material Model**

| 1       | 2    | 3    | 4   | 5    | 6     | 7  | 8     | 9     | 10 |
|---------|------|------|-----|------|-------|----|-------|-------|----|
| MATNLE3 | MID  | REF  | PR  | RHO  | ALPH  | G  | TLIM  | CLIM  |    |
|         |      | TE   | TPR | TRHO | TALPH | TG | TTLIM | TCLIM |    |
|         | KIND | TYPE | LIM | GE   | NAME  |    |       |       |    |

**Format 4: Linear Elasticity with Tension and/or Compression Limits**

| 1       | 2    | 3    | 4   | 5    | 6     | 7  | 8     | 9     | 10 |
|---------|------|------|-----|------|-------|----|-------|-------|----|
| MATNLE4 | MID  | REF  | PR  | RHO  | ALPH  | G  | TLIM  | CLIM  |    |
|         |      | TE   | TPR | TRHO | TALPH | TG | TTLIM | TCLIM |    |
|         | KIND | TYPE | LIM | GE   | NAME  |    |       |       |    |

**Format 5: Bi-Modulus Linear Elasticity with Tension/Compression Limits**

| 1       | 2    | 3    | 4   | 5    | 6     | 7  | 8     | 9     | 10 |
|---------|------|------|-----|------|-------|----|-------|-------|----|
| MATNLE5 | MID  | E    | PR  | RHO  | ALPH  | G  | TLIM  | CLIM  |    |
|         |      | TE   | TPR | TRHO | TALPH | TG | TTLIM | TCLIM |    |
|         | KIND | TYPE | LIM | GE   | NAME  |    |       |       |    |
|         | EC   | PRC  | TEC | TPRC |       |    |       |       |    |

**Format 6: Orthotropic Nonlinear Elasticity based upon strains in local directions**

| 1       | 2   | 3    | 4    | 5    | 6     | 7     | 8     | 9    | 10 |
|---------|-----|------|------|------|-------|-------|-------|------|----|
| MATNLE6 | MID | E11  | E22  | E33  | PR12  | PR23  | PR31  | RHO  |    |
|         |     | TE11 | TE22 | TE33 | TPR12 | TPR23 | TPR31 | TRHO |    |
|         | G12 | G23  | G31  | A11  | A22   | A33   |       |      |    |

|  | TG12 | TG23 | TG31 | TA11 | TA22 | TA33 |  |  |  |
|--|------|------|------|------|------|------|--|--|--|
|  | KIND | TYPE | LIM  | GE   | NAME |      |  |  |  |

**Example - Format 3:**

|         |   |          |      |          |          |  |        |        |  |
|---------|---|----------|------|----------|----------|--|--------|--------|--|
| MATNLE3 | 2 | 2.90E+03 | 0.49 | 1.33E-04 | 2.60E-06 |  | 999999 | 999999 |  |
| +       |   |          |      |          |          |  |        |        |  |
| +       |   | 1        | 1    | 0        | MATNLE2A |  |        |        |  |

**Example - Format 4:**

|         |   |          |      |          |          |  |     |    |  |
|---------|---|----------|------|----------|----------|--|-----|----|--|
| MATNLE4 | 2 | 2.90E+03 | 0.49 | 1.33E-04 | 2.60E-06 |  | 300 | 25 |  |
| +       |   |          |      |          |          |  |     |    |  |
| +       |   | 1        | 1    | 0        | MATNLE4A |  |     |    |  |

**Example - Format 5:**

|         |   |          |      |          |          |  |     |    |  |
|---------|---|----------|------|----------|----------|--|-----|----|--|
| MATNLE5 | 2 | 2.90E+03 | 0.49 | 1.33E-04 | 2.60E-06 |  | 300 | 25 |  |
| +       |   |          |      |          |          |  |     |    |  |
| +       |   | 1        | 1    | 0        | MATNLE4A |  |     |    |  |

**Example - Format 6:**

|         |          |          |          |          |          |          |          |        |  |
|---------|----------|----------|----------|----------|----------|----------|----------|--------|--|
| MATNLE6 | 2        | 2.90E+03 | 2.80E+03 | 0.491    | 0.492    | 0.493    | 1.00E-04 | 999999 |  |
| +       |          |          |          |          |          |          |          |        |  |
| +       | 6.00E+01 | 5.00E+01 | 4.00E+01 | 2.00E-06 | 1.50E-06 | 1.00E-06 |          |        |  |
| +       |          |          |          |          |          |          |          |        |  |
| +       |          | 1        | 1        | 0        | MATNLE6A |          |          |        |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MATNLEi entry. (Integer > 0; no Default)  |
| E          | Young's modulus. (Real; no Default)  |
| PR         | Poisson's ratio. (Real; no Default)  |
| RHO        | Mass density. (Real; no Default)   |
| ALPH       | Coefficient of thermal expansion. (Real; no Default)   |
| G          | Shear Modulus. (Real; no Default)  |
| TLIM       | Tensile stress limit. (Real; no Default)   |
| CLIM       | Compressive stress limit. (Real; no Default)   |
| TE         | ID of TABL3Di entry for E vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required) |

| Descriptor | Meaning  |
|------------|--|
| TPR        | ID of TABL3Di entry for PR vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TRHO       | ID of TABL3Di entry for density vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)   |
| TALPH      | ID of TABL3Di entry for ALPH vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TG         | ID of TABL3Di entry for G vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)   |
| TTLIM      | ID of TABL3Di entry for TLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TCLIM      | ID of TABL3Di entry for CLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| KIND       | Tension/compression usage flag. (Integer; Default = 1)<br>1 = Both tension and compression data will be used<br>2 = Only tension data will be used   |
| TYPE       | Poisson ratio/Bulk Modulus flag. (Integer; Default = 1)<br>1 = Constant Poisson's ratio will be used<br>2 = Constant bulk modulus will be used   |
| LIM        | Stress limit flag. (Integer; Default = 0)<br>0 = Material has no tension or compression limits<br>1 = Material has tension limits<br>2 = Material has compression limits<br>3 = Material has both tension and compression limits |
| GE         | Structural damping coefficient. (Real; Default = 0.0)  |
| NAME       | Material name. (Character; no Default) Optional name of the material, may be blank or as long as 16 characters.  |
| EC         | Young's modulus in compression. (Real; no Default)   |
| PRC        | Poisson's ratio in compression. (Real; no Default)   |
| TEC        | ID of TABL3Di entry for EC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TPRC       | ID of TABL3Di entry for PRC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)   |
| E11        | Orthotropic modulus E11. (Real; no Default)  |
| E22        | Orthotropic modulus E22. (Real; no Default)  |
| E33        | Orthotropic modulus E33. (Real; no Default)  |

| Descriptor | Meaning   |
|------------|---|
| PR12       | Orthotropic Poisson's ratio in 12 direction. (Real; no Default)   |
| PR23       | Orthotropic Poisson's ratio in 23 direction. (Real; no Default)   |
| PR31       | Orthotropic Poisson's ratio in 31 direction. (Real; no Default)   |
| TE11       | ID of TABL3Di entry for E11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TE22       | ID of TABL3Di entry for E22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TE33       | ID of TABL3Di entry for E33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TPR12      | ID of TABL3Di entry for PR12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required) |
| TPR23      | ID of TABL3Di entry for PR23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required) |
| TPR31      | ID of TABL3Di entry for PR31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required) |
| G12        | Orthotropic shear modulus G12. (Real; no Default)   |
| G23        | Orthotropic shear modulus G23. (Real; no Default)   |
| G31        | Orthotropic shear modulus G31. (Real; no Default)   |
| A11        | Orthotropic coefficient of thermal expansion in 1 direction. (Real; no Default)   |
| A22        | Orthotropic coefficient of thermal expansion in 2 direction. (Real; no Default)   |
| A33        | Orthotropic coefficient of thermal expansion in 3 direction. (Real; no Default)   |
| TG12       | ID of TABL3Di entry for G12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TG23       | ID of TABL3Di entry for G23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TG31       | ID of TABL3Di entry for G31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TA11       | ID of TABL3Di entry for A11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TA22       | ID of TABL3Di entry for A22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |
| TA33       | ID of TABL3Di entry for A33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)  |

Remarks:

1. The structural damping constant GE is specified in MARC under the option DAMPING as a numerical damping  $\gamma$ ,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE;  $\alpha$  and  $\beta$  are equivalent to user parameters ALPHA1 and ALPHA2.

2. This Bulk Data entry accommodates MARC's input data under the model definition option NLELAST for material model types 2-6.
3. 3D tables associated with this entry should normally select independent variable 69 (equivalent mechanical strain) or 73 (any strain component) to define the stress-strain curve. Normally the only other independent 3D table variables used with this entry are 12 (temperature) or 16 (equivalent strain rate).
4. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**MATORT****Elastic 3D Orthotropic Material Properties**

Specifies elastic orthotropic material properties. Can be specified for all 3-D and 2-D continuum elements and shells in SOL400/600/700. Can also be specified with solid composite, solid shell composite and pyramid elements in all linear solution sequences between SOL101 and SOL112, and analysis only for SOL200.

**Format:**

| 1      | 2      | 3     | 4     | 5   | 6    | 7    | 8    | 9   | 10 |
|--------|--------|-------|-------|-----|------|------|------|-----|----|
| MATORT | MID    | E1    | E2    | E3  | NU12 | NU23 | NU31 | RHO |    |
|        | G12    | G23   | G31   | A1  | A2   | A3   | TREF | GE  |    |
|        | IYLD   | IHARD | SY    |     | Y1   | Y2   | Y3   | N/A |    |
|        | Yshr1  | Yshr2 | Yshr3 | N/A | N/A  | N/A  | N/A  | N/A |    |
|        | OPTION | FILE  | X1    | Y1  | Z1   | X2   | Y2   | Z2  |    |

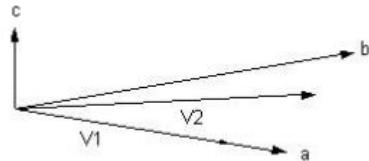
**Example:**

|        |     |      |       |       |      |  |  |  |  |
|--------|-----|------|-------|-------|------|--|--|--|--|
| MATORT | 100 | 3.e6 | 2.8e7 | 1.5e5 | 0.25 |  |  |  |  |
|--------|-----|------|-------|-------|------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MATORT entry. (Integer > 0; No Default)   |
| E1         | Modulus of elasticity in longitudinal or 1-direction. See Remark 1. (Real > 0)   |
| E2         | Modulus of elasticity in lateral direction or 2-direction. (Real > 0; no Default; must be entered)                             |
| E3         | Modulus of elasticity in thickness direction or 3-direction. (Real > 0; no Default; must be entered)                           |
| NU12       | Poisson's ratio $\epsilon_2/\epsilon_1$ for uniaxial loading in 1-direction. See Remark 2. (Real; no Default; must be entered) |
| NU23       | Poisson's ratio $\epsilon_3/\epsilon_2$ for uniaxial loading in 2-direction. (Real; no Default; must be entered)               |
| NU31       | Poisson's ratio $\epsilon_1/\epsilon_3$ for uniaxial loading in 3-direction. (Real; Default = NU23)                            |
| RHO        | Mass density. (Real; Default = 0.0)  |
| G12        | Shear modulus in plane 1-2. See Remark 3. (Real > 0; no Default; must be entered)  |
| G23        | Shear modulus in plane 2-3. (Real > 0.; no Default; must be entered)   |
| G31        | Shear modulus in plane 3-1. (Real > 0; no Default; must be entered)  |
| Ai         | Coefficient of thermal expansion in i-direction (Real; Default = 0.0).   |

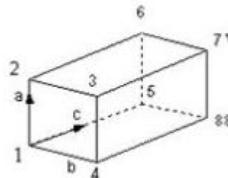
| Descriptor | Meaning  |
|------------|--|
| TREF       | Reference temperature at which the thermal expansion coefficient is measured. TREF is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)   |
| GE         | Structural damping coefficient. (Real; Default = 0.0).   |
| IYLD       | Integer pertaining to one of the following yield criterion: (Integer)<br><br>-1 = Elastic<br>1 = von Mises (Default)<br>2 = Hill (1948) yield<br>3 = Barlat (1991) yield<br>4 = Viscoplasticity through user subroutine UVSCPL |
| IHARD      | Integer pertaining to one of the following work hardening rules: (Integer)<br>1 = Isotropic (Default)<br>2 = Kinematic<br>3 = Combined Isotropic/Kinematic   |
| SY         | Equivalent (von Mises) tensile yield stress. (Real > 0.0 or blank; Default = 0.0)  |
| Y1         | Hill's yield stress ratio in direction 1. (Real > 0.0 or blank; Default = 0.0)   |
| Y2         | Hill's yield stress ratio in direction 2. (Real > 0.0 or blank; Default = 0.0)   |
| Y3         | Hill's yield stress ratio in direction 3. (Real > 0.0 or blank; Default = 0.0)   |
| Yshr1      | Hill's yield shear stress ratio in direction 1. (Real > 0.0 or blank; Default = 0.0)   |
| Yshr2      | Hill's yield shear stress ratio in direction 2. (Real > 0.0 or blank; Default = 0.0)   |
| Yshr3      | Hill's yield shear stress ratio in direction 3. (Real > 0.0 or blank; Default = 0.0)   |

| Descriptor | Meaning   |
|------------|---|
| OPTION     | Material axes option used to determine how the local material axis system is defined. See Remark 7. (Character; default=ELEM)   |
| VECT       | Globally orthotropic with the material axes defined by two vectors V1 and V2, specified using the fields X1, Y1, Z1 and X2, Y2, Z2. The a-axis is defined by the first vector. The b- and c-axes are then defined as: |



#### ELEM

Globally orthotropic material with the material axes defined by element topology. Grid point 1 defines the origin, grid point 5 lies on the c-axis, and grid point 2 lies in the ac-plane. The a, b, and c axes are defined as follows:



#### ELMAT

Orthotropic material properties and the material coordinate system is defined by the element. The material properties are read from a file (formatted). The filename is specified in the sixth field of the first line.

Format of material properties file:

```
Record#
EID, DENSITY, DUMMY, DUMMY, DUMMY,
E1, E2, E3, G12, G23, G31,
NU12, NU13, NU23, NU21, NU31, NU32
```

#### ELPROP

Globally orthotropic material with the material axes defined by element topology (see also ELEM). The elasticity matrix is available per element.

| Descriptor | Meaning   |
|------------|---|
| FILE       | Material file name (OPTION=ELMAT only). See Remark 7.<br>(Character; default=blank)           |
| X1, Y1, Z1 | Components of the vector V1 in the basic coordinate system. See Remark 7. (Real; default=0.0) |
| X2, Y2, Z2 | Components of the vector V2 in the basic coordinate system. See Remark 7. (Real; default=0.0) |

**Remarks:**

1. All the material constants are specified in the orthotropic material coordinates in 1, 2, and 3 direction.
2. In general,  $v_{12}$  is not the same as  $v_{21}$ , but they are related by  $v_{ij}/E_i = v_{ji}/E_j$ . Furthermore, material stability requires that

$$E_i > v_{ij}^2 E_j$$

$$\text{and } 1 - v_{12}v_{21} - v_{23}v_{32} - v_{31}v_{13} - 2v_{21}v_{32}v_{13} > 0.$$

3. It may be difficult to find all nine orthotropic constants. In some practical problems, the material properties may be reduced to normal anisotropy in which the material is isotropic in a plane, e.g., in plane 1-2 and has different properties in the direction normal to the plane 1-2. In the plane of isotropy, the properties are reduced to

$$E_1 = E_2 = E_p$$

$$v_{31} = v_{32} = v_{np}$$

$$v_{13} = v_{23} = v_{pn}$$

$$G_{13} = G_{23} = G_n$$

$$\text{with } v_{np}/E_n = v_{pn}/E_p \text{ and } G_p = \frac{E_p}{2(1 + v_p)}.$$

There are five independent material constants for normal anisotropy (e.g.,  $E_p$ ,  $E_n$ ,  $v_p$ ,  $v_{np}$ ,  $G_n$ ).

In case the material has a planar anisotropy, in which the material is orthotropic only in a plane, the elastic constants are reduced to seven, e.g.,  $E_1$ ,  $E_2$ ,  $E_3$ ,  $v_{12}$ ,  $G_{12}$ ,  $G_{23}$ ,  $G_{31}$ .

4. If  $Y2$  and/or  $Y3$  are blank, then  $Y1$  is assumed if entered. If  $Ysh2$  and/or  $Ysh3$  are blank,  $Ysh1$  is assumed if entered.
5. Do not enter values for  $SY$ ,  $Y1$ ,  $Y2$ ,  $Y3$ ,  $YSHR1$ ,  $YSHR2$  or  $YSHR3$  unless plasticity is to be taken into account.

6. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
7. OPTION, FILE, X1, Y1, Z1, X2, Y2 and Z2 fields are only supported by new SOL700.
8. New SOL700 does not support A1, A2, A3, TREF, GE, IYLD, IHARD, SY, Y1, Y2, Y3, Yshr1, Yshr2 and Yshr3 fields.

Notes:

1. The structural damping constant GE is specified as a numerical damping  $\gamma$ , i.e.,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE.

2. This Bulk Data entry accommodates Marc's input data under the model definition option ORTHOTROPIC.

**MATPE1****Isotropic Poroelastic Material Property definition**

Defines the material properties for an isotropic poroelastic material.

**Format:**

| 1      | 2    | 3     | 4       | 5    | 6   | 7   | 8   | 9   | 10 |
|--------|------|-------|---------|------|-----|-----|-----|-----|----|
| MATPE1 | MID  | MAT1  | MAT10   | BIOT |     |     |     |     |    |
|        | VISC | GAMMA | PRANDTL | POR  | TOR | AFR | VLE | TLE |    |

**Example:**

|        |        |     |        |       |     |       |        |        |  |
|--------|--------|-----|--------|-------|-----|-------|--------|--------|--|
| MATPE1 | 101    | 1   | 10     |       |     |       |        |        |  |
|        | 1.84-8 | 1.4 | 7.13-1 | 9.5-1 | 1.4 | 2.5-5 | 9.32-2 | 9.32-2 |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material identification number, referenced by PSOLID entry. (Integer>0)   |
| MAT1       | Identification number of MAT1 (and MATF1 if it is frequency-dependent) entry for the skeleton (solid-phase) of porous material. (Integer>0, Required) |
| MAT10      | Identification number of MAT10 entry for the fluid-phase of porous material. (Integer>0, Required)  |
| BIOT       | Biot factor. (Real>0.0, Default=1.0)  |
| VISC       | Fluid dynamic viscosity. See Remark 4. (Real>0.0, Required)   |
| GAMMA      | Fluid ratio of specific heats. See Remark 5. (Real>0.0, Default=1.402)  |
| PRANDTL    | Fluid Prandtl number, Pr. See Remark 6. (Real>0.0, Default=0.71)  |
| POR        | Porosity of porous material. (Real>0.0, Required)   |
| TOR        | Tortuosity of porous material. (Real>0.0, Default=1.0)  |
| AFR        | Air flow resistivity. (Real>0.0, Required)  |
| VLE        | Viscous characteristic length. (Real>0.0, Required)   |
| TLE        | Thermal characteristic length. (Real>0.0, Required)   |

**Remarks:**

1. MATPE1 entry is referenced by PSOLID cards when FCTN=PORO. It is used to define the isotropic poroelastic material properties for both solid and fluid phases of a poroelastic medium.
2. MATPE1 entry must be put under BEGIN BULK TRMC.
3. For a frequency-dependent skeleton material, MATF1 specifies the material properties at frequencies on matching MAT1 fields.
4. Fluid dynamic viscosity, VISC, is temperature dependent. At 15°C (Celsius), its value is  $1.82 \times 10^{-6}$  N.s/m<sup>2</sup> in the SI units.

5. GAMMA is defined as  $C_p/C_v$  where  $C_p$  is the specific heat at constant pressure and  $C_v$  the specific heat at constant volume. Its default value is 1.402 for the air at ambient temperature.
6. PRANDTL number, Pr, is defined as  $\mu C_p/k$  where  $\mu$  is the fluid dynamic viscosity (VISC) and k the thermal conductivity. For the dry air at ambient temperature, its value is around 0.7-0.8.
7. Porosity,  $\Omega$ , is defined as  $V_f/V_t$  where  $V_f$  is the volume of fluid and  $V_t$  is the total volume of the poroelastic material. Hence, porosity is dimensionless.
8. Tortuosity measures the complexity of the path an air particle must follow to process from one point to another point. It is a dimensionless parameter.
9. Resistivity measures the resistance of poroelastic material against the flow of fluid. In SI unit, it is measured in N.s/m<sup>4</sup>.
10. The density of poroelastic material is defined as  $\rho_{pem} = \Omega\rho_f + (1 - \Omega)\rho_s$  where  $\rho_f$  is the fluid density from MAT10 and  $\rho_s$  is the solid density from MAT1.
11. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**MATRIG****Rigid Material**

Defines the properties of a rigid material in SOL 700.

**Format:**

| 1      | 2        | 3        | 4        | 5   | 6    | 7   | 8   | 9  | 10 |
|--------|----------|----------|----------|-----|------|-----|-----|----|----|
| MATRIG | MID      | RHO      | E        | NU  | MASS | XC  | YC  | ZC |    |
|        | IXX      | IXY      | IXZ      | IYY | IYZ  | IZZ | CID |    |    |
|        | VX       | VY       | VZ       | WX  | WY   | WZ  |     |    |    |
|        | XC-LOCAL | YC-LOCAL | ZC-LOCAL |     |      |     |     |    |    |

**Example:**

|        |      |       |        |      |      |      |     |      |  |
|--------|------|-------|--------|------|------|------|-----|------|--|
| MATRIG | 7    | 7850. | 210.E9 | 0.3  | 750  | 0.0  | 7.0 | -3.0 |  |
|        | 17.0 | 13.2  | 14.3   | 20.9 | 15.7 | 10.0 | 12  |      |  |
|        |      |       | 13.3   |      |      |      |     |      |  |

| Descriptor                       | Meaning   | Type        | Default                  |
|----------------------------------|---|-------------|--------------------------|
| MID                              | Unique material number.   | I > 0       | Required                 |
| RHO                              | Density   | R > 0       | 1.0                      |
| E                                | Young's modulus   | R > 0       | 1.0                      |
| NU                               | Poisson's ratio   | 0.0 R < 0.5 | 0.2                      |
| MASS                             | Mass of the rigid body.   | R > 0.0     | See Remark 2.            |
| XC, YC, ZC                       | x, y, and z coordinates of the center of gravity.   | R           | See Remark 6.            |
| IXX, IXY, IXZ,<br>IYY, IYZ, IZZ  | Inertia tensor of the rigid body about the center of gravity.   | R           | See Remark 6.            |
| CID                              | Number of a coordinate system in which the inertia tensor and the center of gravity are defined.            | I > 0       | See Remarks 7.<br>and 8. |
| VX, VY, VZ                       | Initial translational velocity of the center of gravity in the basic coordinate system.                     | R           | 0.0                      |
| WX, WY, WZ                       | Initial rotational velocities of the rigid body about the center of gravity in the basic coordinate system. | R           | 0.0                      |
| XC-LOCAL<br>YC-LOCAL<br>ZC-LOCAL | x, y, and z local coordinates of the center of gravity  | R           | See Remark 8.            |

**Remarks:**

1. All coordinates are defined in the basic coordinate system.

2. If MASS is blank or zero, the mass will be calculated from the density and the geometry of the mesh defining the rigid body.
3. The continuation lines are not required.
4. The MATRIG definition is used instead of a material definition and is referenced by properties PSOLIDn, PSHELLn, PBAR, and PBEAMn. Different properties can refer to the same MATRIG entry forming one rigid body.
5. If the fields VX, VY, VZ, WX, WY, and WZ are blank, then the initial conditions of the rigid body are calculated from the initial velocities on the TIC and TIC1 entries referring to grid points attached to the rigid body. The net initial conditions are the average of those for all the grid points attached to the rigid body.  
If the initial conditions are set using the VX, VY, VZ, WX, WY, and WZ fields, the TIC and TIC1 entries referring to grid points attached to the rigid body are ignored.
6. If the inertia tensor or the coordinates of the center of gravity are undefined, then they will be computed from the density or mass and the geometry of the mesh defining the rigid body.
7. The inertia tensor can only be defined in a local rectangular coordinate system. If the entry for a local coordinate system is left blank, then the inertia tensor is defined in the global coordinate system.
8. The center of gravity can be defined in a local rectangular coordinate system (CID). However, XC YC ZC (x, y, and z coordinates of the center of gravity in the basic coordinate system) should be left blank when XC-LOCAL YC-LOCAL ZC-LOCAL (x, y, and z coordinates of the center of gravity in a local coordinate system) defined.

**MATS1****Material Stress Dependence**

Specifies stress-dependent material properties for use in applications involving nonlinear materials. This entry is used if a MAT1 entry is specified with the same MID in a nonlinear solution sequence (SOLs 106, 129 and 400). For SOL 400 the MATEP option is the recommended approach for elastic-plastic materials.

**Format:**

| 1     | 2   | 3   | 4    | 5 | 6  | 7  | 8      | 9      | 10 |
|-------|-----|-----|------|---|----|----|--------|--------|----|
| MATS1 | MID | TID | TYPE | H | YF | HR | LIMIT1 | LIMIT2 |    |

**Example:**

|       |    |    |         |     |   |   |      |  |  |
|-------|----|----|---------|-----|---|---|------|--|--|
| MATS1 | 17 | 28 | PLASTIC | 0.0 | 1 | 1 | 2.+4 |  |  |
|-------|----|----|---------|-----|---|---|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MAT1 entry. (Integer > 0)   |
| TID        | Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark 3. (Integer $\geq 0$ or blank)   |
| TYPE       | Type of material nonlinearity. See Remarks. (Character: "NLELAST" for nonlinear elastic or "PLASTIC" for elastoplastic.)   |
| H          | Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, H = 0.0. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark 2. (Real) |
| YF         | Yield function criterion, selected by one of the following values (Integer):<br>1 von Mises (Default)<br>2 Tresca<br>3 Mohr-Coulomb<br>4 Drucker-Prager  |
| HR         | Hardening Rule, selected by one of the following values (Integer):<br>1 Isotropic (Default)<br>2 Kinematic<br>3 Combined isotropic and kinematic hardening   |
| LIMIT1     | Initial yield point. See <a href="#">Table 9-25</a> . (Real)   |
| LIMIT2     | Internal friction angle, measured in degrees, for the Mohr-Coulomb and Drucker-Prager yield criterion. See <a href="#">Table 9-25</a> . (0.0 $\leq$ Real $<$ 45.0)   |

Table 9-25 Yield Functions Versus LIMIT1 and LIMIT2

| Yield Function (YF)                       | LIMIT1   | LIMIT2   |
|---|--|--|
| von Mises (1)<br>or Tresca (2)            | Initial Yield Stress in Tension, $Y_1$         | Not used                                       |
| Mohr-Coulomb (3)<br>or Drucker-Prager (4) | $2^* \text{Cohesion}, 2c$ (in units of stress) | Angle of Internal Friction $\phi$ (in degrees) |

## Remarks:

1. If TYPE = "NLELAST", then MID may refer to a MAT1 entry only. Also, the stress-strain data given in the TABLES1 entry will be used to determine the stress for a given value of strain. The values H, YF, HR, LIMIT1, and LIMIT2 will not be used in this case.

Thermoelastic analysis with temperature-dependent material properties is available for linear and nonlinear elastic isotropic materials (TYPE = "NLELAST") and linear elastic anisotropic materials. Four options of constitutive relations exist. The relations appear in [Table 23](#) along with the required Bulk Data entries.

Table 23 Constitutive Relations and Required Material Property Entries

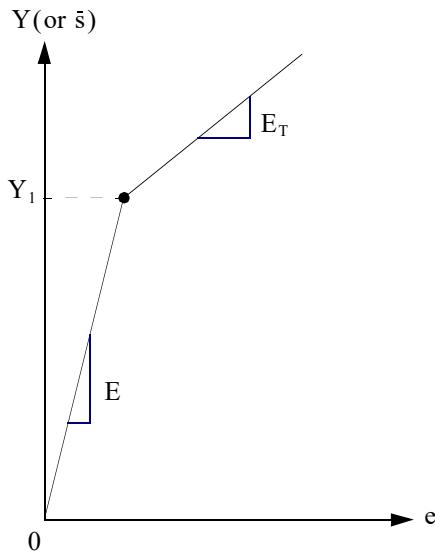
| Constitutive Relation   | Required Bulk Data Entries               |
|---|--|
| $\{\sigma\} = [G_e(T)]\{\varepsilon\}$  | MAT1 and MATT1                           |
| $\{\sigma\} = \frac{\bar{E}(\sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$    | MAT1, MATT1, MATS1, and TABLES1          |
| $\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e]\{\varepsilon\}$    | MAT1, MATS1, TABLEST, and TABLES1        |
| $\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$ | MAT1, MATT1, MATS1, TABLEST, and TABLES1 |

In [Table 23](#)  $\{\sigma\}$  and  $\{\varepsilon\}$  are the stress and strain vectors,  $[G_e]$  the elasticity matrix,  $\bar{E}$  the effective elasticity modulus, and  $E$  the reference elasticity modulus.

2. If TYPE = "PLASTIC", the elastic stress-strain matrix is computed from MAT1 entry, and then the isotropic plasticity theory is used to perform the plastic analysis. In this case, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The plasticity modulus ( $H$ ) is related to the tangential modulus ( $E_T$ ) by

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where  $E$  is the elastic modulus and  $E_T = dY/d\varepsilon$  is the slope of the uniaxial stress-strain curve in the plastic region. See [Figure 9-111](#).



[Figure 9-111 Stress-Strain Curve Definition When H Is Specified in Field 5](#)

3. If TID is given, TABLES1 entries  $(X_i, Y_i)$  of stress-strain data  $(\varepsilon_k, Y_k)$  must conform to the following rules (see [Figure 9-112](#)):
  - If TYPE = "PLASTIC", the stress-total strain curve (i.e., TYPE in TABLES1 must be 1) must be defined in the first quadrant. The first point must be at the origin  $(X_1 = 0, Y_1 = 0)$  and the second point  $(X_2, Y_2)$  must be at the initial yield point  $(Y_1$  or  $2c)$  specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the value of  $E$ . Also, TID may not reference a TABLEST entry.

The stress-total strain curve has 2 different definitions

- For conventional element, small strain formulation in terms of engineering strain and stress is used. Therefore, engineering stress-strain curve should be used here.
- For advanced nonlinear element, large strain formulation in terms of logarithmic strain and true stress is adopted. Then true stress-logarithmic strain curve should be used.

Note that SOL 106 and SOL 129 supports conventional element only. SOL 400 supports both of the conventional and advanced elements.

For TYPE = "PLASTIC", it should be noted that for conventional elements, strains should be below .05 to small strain formulation. Within this strain range, difference between logarithmic strain and engineering strain may be negligible. If the element strains exceed .05, then it is recommended to use the MATEP entry which will automatically switch to a large strain formulation in SOL 400

For SOL 600, if use of the MATS1 entry is desired, then it should be used in conjunction with PARAM,MRTABLS1,value.

- If TYPE = "NLELAST", the full stress-strain curve ( $-\infty < x < \infty$ ) may be defined in the first and the third quadrant to accommodate different uniaxial compression data. If the curve is defined only in the first quadrant, then the curve must start at the origin (X1 = 0.0, Y = 0.0) and the compression properties will be assumed identical to tension properties.

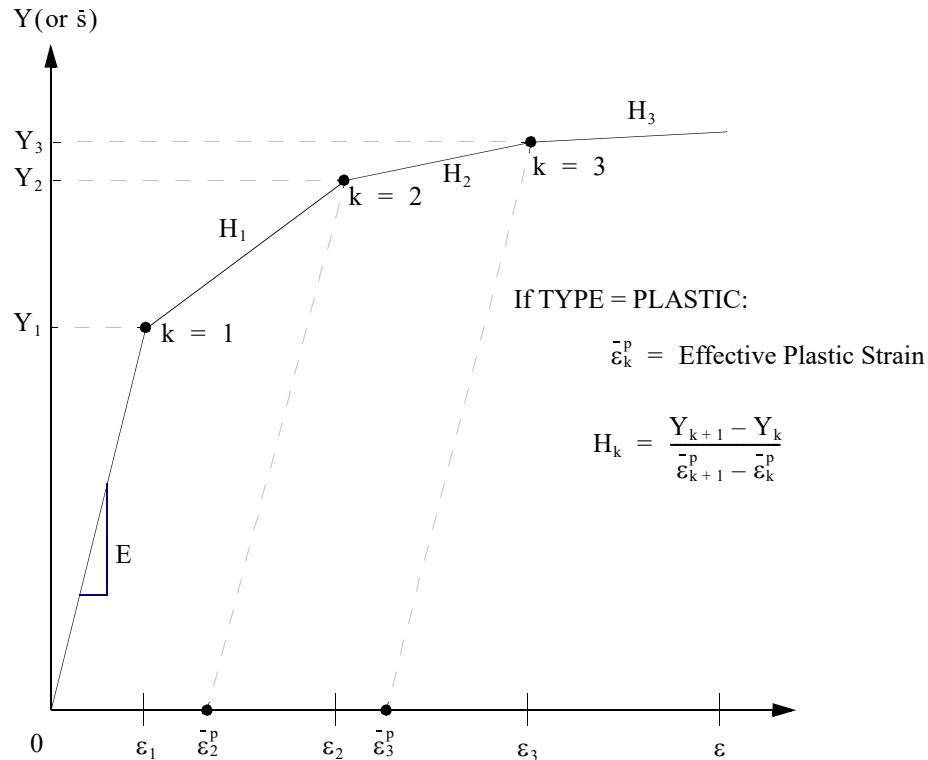


Figure 9-112 Stress-Strain Curve Definition When TID Is Specified in Field 3

4. In SOL 400, MATS1 is supported for 2 sets of elements:

- a. Elements with no property extensions that are prescribed through PROD, PBEAML, PSHELL and PSOLID. The original small-strain element formulations that are also available in SOL106 are used in these cases.
- b. Large-strain elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for TYPE = PLASTIC, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral. Note also that, prior to MD Nastran 2010, if the property extensions were missing, then for supported elements (i.e., PROD, PBEAML, PSHELL or PSOLID), the analysis would use the element formulations in (a), or ignore the MATS1 data for unsupported elements. From MD Nastran 2010 onwards, if the property extensions are missing, then they are automatically added with appropriate settings by the program unless the element belongs to group (a) or the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATS3**

## Advanced Orthotropic, Nonlinear Elastic Materials for Axisymmetric Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions in SOL 400.

**Format:**

| 1     | 2   | 3   | 4    | 5   | 6      | 7      | 8     | 9    | 10 |
|-------|-----|-----|------|-----|--------|--------|-------|------|----|
| MATS3 | MID | TEX | TETH | TEZ | TNUXTH | TNUTHZ | TNUZX | TRHO |    |
|       |     |     | TGZX | TAX | TATH   | TAZ    |       |      |    |

**Example:**

|       |    |    |    |  |    |  |  |  |  |
|-------|----|----|----|--|----|--|--|--|--|
| MATS3 | 33 | 56 |    |  | 67 |  |  |  |  |
|       |    |    | 12 |  |    |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MAT3 entry. (Integer > 0; no Default)                           |
| TEX        | ID of TABL3Di entry for EX. (Integer; no Default; leave blank if table is not required)    |
| TETH       | ID of TABL3Di entry for ETH. (Integer; no Default; leave blank if table is not required)   |
| TEZ        | ID of TABL3Di entry for EZ. (Integer; no Default; leave blank if table is not required)    |
| TNUXTH     | ID of TABL3Di entry for NUXTH. (Integer; no Default; leave blank if table is not required) |
| TNUTHZ     | ID of TABL3Di entry for NUTHZ. (Integer; no Default; leave blank if table is not required) |
| TNUZX      | ID of TABL3Di entry for NUZX. (Integer; no Default; leave blank if table is not required)  |
| TRHO       | ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)   |
| TGZX       | ID of TABL3Di entry for GZX. (Integer; no Default; leave blank if table is not required)   |
| TAX        | ID of TABL3Di entry for AX. (Integer; no Default; leave blank if table is not required)    |
| TATH       | ID of TABL3Di entry for ATH. (Integer; no Default; leave blank if table is not required)   |
| TAZ        | ID of TABL3Di entry for AZ. (Integer; no Default; leave blank if table is not required)    |

**Remarks:**

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc.

2. In SOL 400, MATS3 is used in conjunction with MAT3 and is only supported for axisymmetric elements with property extensions. This implies that for such elements, PLPLANE should be associated with PSHLN2. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS3 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATS8**

## Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements in SOL 400.

**Format:**

| 1     | 2   | 3   | 4   | 5     | 6    | 7    | 8    | 9    | 10 |
|-------|-----|-----|-----|-------|------|------|------|------|----|
| MATS8 | MID | TE1 | TE2 | TNU12 | TG12 | TG1Z | TG2Z | TRHO |    |
|       | TA1 | TA2 |     |       |      |      |      |      |    |

**Example:**

|       |    |    |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|
| MATS8 | 34 | 12 |  |  |  |  |  |  |  |
|       | 1  |    |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a MAT8 entry. (Integer > 0; no Default)                          |
| TE1        | ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)   |
| TE2        | ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)   |
| TNU12      | ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required) |
| TG12       | ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)  |
| TG1Z       | ID of TABL3Di entry for G1Z. (Integer; no Default; leave blank if table is not required)  |
| TG2Z       | ID of TABL3Di entry for G2Z. (Integer; no Default; leave blank if table is not required)  |
| TRHO       | ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)  |
| TA1        | ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)   |
| TA2        | ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)   |

**Remarks:**

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. As an example, say the analysis required TE1 to be a function of strain. Then the corresponding TABL3D0 entry is TABL3D0, KIND=73 to specify *local strain component* as the independent variable. The dependent variable is *TE1*.
2. In SOL 400, MATS8 is used in conjunction with MAT8 and is only supported for shell elements with property extensions. This implies that for such elements, PSHELL should be associated with PSHLN1. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS8 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATSMA****Material Properties for Shape Memory Alloys**

For SOL 600 and SOL 400 only.

**Format:**

| 1              | 2                 | 3                             | 4                             | 5              | 6                           | 7                            | 8                | 9 | 10 |
|----------------|-------------------|-------------------------------|-------------------------------|----------------|-----------------------------|------------------------------|------------------|---|----|
| MATSMA         | MID               | MODEL                         | T <sub>0</sub>                | L              |                             |                              |                  |   |    |
| E <sub>a</sub> | v <sub>a</sub>    | α <sub>a</sub>                | σ <sub>a</sub>                | ρ <sub>a</sub> | σ <sub>s</sub> <sup>A</sup> | σ <sub>f</sub> <sup>AS</sup> | C <sub>a</sub>   |   |    |
| E <sub>m</sub> | v <sub>m</sub>    | α <sub>m</sub>                | σ <sub>m</sub>                | ρ <sub>m</sub> | σ <sub>s</sub> <sup>S</sup> | σ <sub>f</sub> <sup>SA</sup> | C <sub>m</sub>   |   |    |
| v <sup>T</sup> | M <sub>frac</sub> | σ <sub>eff</sub> <sup>g</sup> | σ <sub>max</sub> <sup>g</sup> |                |                             |                              |                  |   |    |
| g <sub>0</sub> | g <sub>a</sub>    | g <sub>b</sub>                | g <sub>c</sub>                | g <sub>d</sub> | g <sub>e</sub>              | g <sub>f</sub>               | g <sub>max</sub> |   |    |

**Example:**

|        |        |      |         |         |    |      |      |      |  |
|--------|--------|------|---------|---------|----|------|------|------|--|
| MATSMA | 1      | 2    | 77.     | 0.05    |    |      |      |      |  |
|        | 50000. | 0.33 | 0.00001 | 1.0E+20 |    | 520. | 600. | 8.66 |  |
|        | 50000. | 0.33 | 0.00001 | 1.0e+20 |    | 300  | 200. | 6.66 |  |
|        | 0.0    | 0.0  | 100.    | 1.0e+20 |    |      |      |      |  |
|        | 300.   | -4.  | 2.      | 0.      | 0. | 0.   | 3.0  | 1.0  |  |

| Descriptor     | Meaning  |
|----------------|--|
| MID            | Material ID. (Integer > 0)   |
| MODEL          | Flag to indicate the model being used. (Integer > 0; see Remark 1.):<br>= 1 (Mechanical: Aruchchio's model)<br>= 2 (Thermo-Mechanical: Asaro-Sayeedvafa model)   |
| T <sub>0</sub> | Reference temperature used to measure stresses. (Real > 0)   |
| L              | For the mechanical model, the parameter L represents maximum deformation, obtainable by detwinning of multiple-variant martensite. (Real > 0; for the thermo-mechanical model, see Remark 2.) (typically 0.06-0.104) |

Austenite Properties (all Real):

|                              |   |
|------------------------------|---|
| E <sub>a</sub>               | Young's modulus of elasticity (typically 60-83 GPa).  |
| v <sub>a</sub>               | Poisson's ratio (typically 0.33).   |
| α <sub>a</sub>               | Coefficient of thermal expansion (typically $3.67 \times 10^{-6} / ^\circ F$ ).   |
| σ <sub>a</sub>               | Equivalent von Mises stress (not used in mechanical model) (typically 195-690 MPa).   |
| ρ <sub>a</sub>               | Mass density.   |
| σ <sub>s</sub> <sup>AS</sup> | Material parameter representing start of Austenite to Martensite transformation.<br>For thermo-mechanical model, the program calculates the Austenite start temperature in the stress-free configuration (A <sub>s</sub> <sup>0</sup> ) related to σ <sub>s</sub> <sup>AS</sup> from relations shown in Table 9-26. |

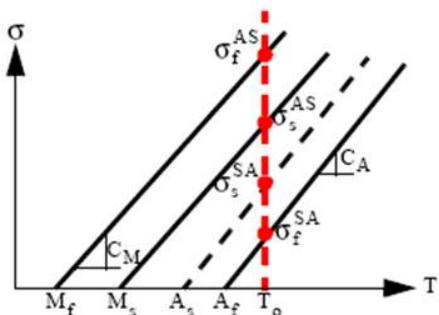
| Descriptor  | Meaning  |
|---|--|
| $\sigma_f^{AS}$   | Material parameter representing finish of Austenite to Martensite transformation. For thermo-mechanical model, the program calculates the Austenite finish temperature in the stress-free configuration ( $A_f^0$ ) related to $\sigma_f^{AS}$ from relations shown in <a href="#">Table 9-26</a> .  |
| $C_a$   | Slope of the stress-dependence of austenite start finish and start temperatures (typically 6-8 MPa).   |
| Martensite Properties (all Real):   |  |
| $E_m$   | Young's modulus of elasticity  |
| $v_m$   | Poisson's ratio  |
| $\alpha_m$  | Coefficient of thermal expansion   |
| $\sigma_m$  | Equivalent von Mises stress (not used in mechanical model)   |
| $\rho_m$  | Density  |
| $\sigma_s^{SA}$   | Material parameter representing start of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite start temperature in the stress-free configuration ( $M_s^0$ ) related to $\sigma_s^{SA}$ from relations shown in <a href="#">Table 9-26</a> .   |
| $\sigma_f^{SA}$   | Material parameter representing finish of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite finish temperature in the stress-free configuration ( $M_f^0$ ) related to $\sigma_f^{SA}$ from relations shown in <a href="#">Table 9-26</a> . |
| $C_m$   | Slope of the stress-dependence of austenite start finish and start temperatures. (typically 5-6 MPa)   |
| The following quantities are applicable to only thermo-mechanical model (all Real): |  |
| $v^T$   | Equivalent volumetric transformation strain. (typically 0.0-0.003)   |
| $M_{frac}$  | Initial martensite volume fraction. (0.0-1.0)  |
| $\sigma_{eff}^g$  | Twinning stress (see Remark 2.). (typically 100-150 MPa)   |
| $\sigma_{max}^g$  | Stress at which the maximum value of $g = g_{max}$ is reached if a cut-off value is needed (normalized with $g_0$ ).   |
| $g_0$   | Stress level used to nondimensionalizing the stress in the function. (2.0-10.0* $\sigma_{eff}^g$ )   |
| $g_a$   | $g$ function coefficient (typically $g_a < 0.0$ )  |
| $g_b$   | $g$ function exponent (typically $g_b = 2.0$ )   |
| $g_c$   | $g$ function coefficient (typically $g_c \geq 0.0$ )   |
| $g_d$   | $g$ function exponent (typically $g_d = 2.25 \sim 2.75$ )  |
| $g_e$   | $g$ function coefficient (typically $g_e \leq 0.0$ )   |
| $g_f$   | $g$ function exponent (typically $g_f = 3.0$ )   |
| $g_{max}$   | Maximum value of function $g$ if a cut-off value is needed (typically $g_{max} = 1.0$ ).   |

**Remarks:**

1. The mechanical (Auricchio's) model can be obtained from the thermo-mechanical model by ignoring the last two rows in the input.
2. Twinning becomes active when the equivalent stress reaches twinning stress. For thermo-mechanical model, the “unstressed transformation temperatures” for Martensite and Austenite,  $M_s^0$ ,  $M_f^0$ ,  $A_s^0$ ,  $A_f^0$  are calculated from the reference temperature, the material parameters representing the start and finish of the Austenite and Martensite transformations, i.e.,  $\sigma_s^{SA}$ ,  $\sigma_f^{SA}$ ,  $\sigma_s^{AS}$ , and  $\sigma_f^{AS}$  as well as the coefficients,  $C_m$ ,  $C_a$  that provide the stress dependence of the transformation temperatures as shown in [Table 9-26](#).

**Table 9-26**

| The Relationship between Mechanical Model and Thermo-Mechanical Model |
|---|
| $\sigma_s^{AS} = (T_o - M_s^0)C_m$                                    |
| $\sigma_f^{AS} = (T_o - M_f^0)C_m$                                    |
| $\sigma_s^{SA} = (T_o - A_s^0)C_a$                                    |
| $\sigma_f^{SA} = (T_o - A_f^0)C_a$                                    |

**Figure 9-113**

3. For the thermo-mechanical model, the equivalent deviatoric strain, (typically 0.05 - 0.085) is automatically calculated by the program as  $eq^T = \sqrt{2/3} * L$  (since it is assumed that the input to this model is an extension of the mechanical model and conversion is done wherever applicable as in [Table 9-26](#). However, in the case where the thermal-mechanical model input parameters are directly used then one must enter a value of  $\sqrt{2/3} * eq$  so that a correct value of  $eq$  is used in the calculations.

4. The Mechanical (Auricchio's) model is not available for 1-D or plane stress conditions. The thermo-mechanical model must be used.
5. In SOL 400, MATSMA is only supported for nonlinear elements with property extensions, PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MSC Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
6. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
7. In SOL 400; remark 4. implies that the Mechanical model may not be used with PBAR, PBARL, PBEAM, PBEAML, PROD, PSHEAR, PSHELL, PCOMP(G). It can only be used with PLPLANE/PSHLN2 (without BEH=PSTRS) and PSOLID/PSLDN1.

**MATSORT**

## Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials in SOL 400.

**Format:**

| 1       | 2    | 3    | 4    | 5   | 6     | 7     | 8     | 9    | 10 |
|---------|------|------|------|-----|-------|-------|-------|------|----|
| MATSORT | MID  | TE1  | TE2  | TE3 | TNU12 | TNU23 | TNU31 | TRHO |    |
|         | TG12 | TG23 | TG31 | TA1 | TA2   | TA3   |       |      |    |

**Example:**

|         |     |    |    |    |    |    |  |  |  |
|---------|-----|----|----|----|----|----|--|--|--|
| MATSORT | 689 | 77 | 77 | 77 |    |    |  |  |  |
|         |     |    |    | 89 | 89 | 89 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a MATORT entry. (Integer > 0; no Default)                        |
| TE1        | ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)   |
| TE2        | ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)   |
| TE3        | ID of TABL3Di entry for E3. (Integer; no Default; leave blank if table is not required)   |
| TNU12      | ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required) |
| TNU23      | ID of TABL3Di entry for NU23. (Integer; no Default; leave blank if table is not required) |
| TNU31      | ID of TABL3Di entry for NU31. (Integer; no Default; leave blank if table is not required) |
| TRHO       | ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)  |
| TG12       | ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)  |
| TG23       | ID of TABL3Di entry for G23. (Integer; no Default; leave blank if table is not required)  |
| TG31       | ID of TABL3Di entry for G31. (Integer; no Default; leave blank if table is not required)  |
| TA1        | ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)   |
| TA2        | ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)   |
| TA3        | ID of TABL3Di entry for A3. (Integer; no Default; leave blank if table is not required)   |

**Remarks:**

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. If one of the independent variables is temperatures then you should not use MATTORT.

2. In SOL 400, MATSORT is used in conjunction with MATORT and is only supported for 2D and 3D elements with property extensions. This implies that for such elements, PCOMP / PCOMPG, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PSHLN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATT1****Isotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.

**Format:**

| 1     | 2     | 3     | 4     | 5     | 6      | 7    | 8 | 9     | 10 |
|-------|-------|-------|-------|-------|--------|------|---|-------|----|
| MATT1 | MID   | T(E)  | T(G)  | T(NU) | T(RHO) | T(A) |   | T(GE) |    |
|       | T(ST) | T(SC) | T(SS) |       |        |      |   |       |    |

**Example:**

|       |    |    |    |    |  |    |  |  |  |
|-------|----|----|----|----|--|----|--|--|--|
| MATT1 | 17 | 32 | 65 | 64 |  | 15 |  |  |  |
|       | 52 |    |    |    |  |    |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)                              |
| T(E)       | Identification number of a TABLEMi entry for the Young's modulus. (Integer $\geq 0$ or blank)  |
| T(G)       | Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)  |
| T(NU)      | Identification number of a TABLEMi entry for the Poisson's ratio. (Integer $\geq 0$ or blank)  |
| T(RHO)     | Identification number of a TABLEMi entry for the mass density. (Integer $\geq 0$ or blank)   |
| T(A)       | Identification number of a TABLEMi entry for the thermal expansion coefficient. See Remark 4. (Positive or negative integer, 0 or blank) |
| T(GE)      | Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or blank)  |
| T(ST)      | Identification number of a TABLEMi entry for the tension stress limit. (Integer $\geq 0$ or blank)                                       |
| T(SC)      | Identification number of a TABLEMi entry for the compression limit. (Integer $\geq 0$ or blank)  |
| T(SS)      | Identification number of a TABLEMi entry for the shear limit. (Integer $\geq 0$ or blank)  |

**Remarks:**

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E is modified by TABLEMi 32, G is modified by TABLEMi 65, NU is modified by TABLEMi 84, A is modified by TABLEMi 15, and ST is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT1 entry.
2. Any quantity modified by this entry must have a value on the MAT1 entry. Initial values of E, G, or NU may be supplied according to Remark 3 on the MAT1 entry.
3. Table references must be present for each item that is temperature dependent. If any one of Young's modulus, shear modulus or Poisson's ratio is temperature dependent, then T(E), T(G) and T(NU) should be specified together to get accurate results.
4. The T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain  $\varepsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative ID<sub>T(A)</sub> value will be changed to |ID<sub>T(A)</sub>| + 100000000 .
5. The continuation entry is not used by SOL 600. For SOL 600, see MATTEP.
6. T(G), T(RHO) and T(GE) are not used by SOL 600. T(E) and T(NU) should be used instead of T(E) and T(G).
7. This entry is not used by SOL 700.

**MATT2****Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

**Format:**

| 1     | 2     | 3       | 4       | 5       | 6       | 7       | 8       | 9      | 10 |
|-------|-------|---------|---------|---------|---------|---------|---------|--------|----|
| MATT2 | MID   | T(G11)  | T(G12)  | T(G13)  | T(G22)  | T(G23)  | T(G33)  | T(RHO) |    |
|       | T(A1) | T(A2)   | T(A3)   |         | T(GE)   | T(ST)   | T(SC)   | T(SS)  |    |
|       |       | T(GE11) | T(GE12) | T(GE13) | T(GE22) | T(GE23) | T(GE33) |        |    |

**Example:**

|       |    |    |  |  |  |  |    |  |  |
|-------|----|----|--|--|--|--|----|--|--|
| MATT2 | 17 | 32 |  |  |  |  | 15 |  |  |
|       |    | 62 |  |  |  |  |    |  |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on a MAT2 entry. (Integer > 0)                            |
| T(Gij)     | Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer $\geq 0$ or blank)                      |
| T(RHO)     | Identification number of a TABLEMk entry for the mass density. (Integer $\geq 0$ or blank)   |
| T(Ai)      | Identification number of a TABLEMk entry for the thermal expansion coefficient. See Remark 3. (Positive or negative integer, 0 or blank) |
| T(GE)      | Identification number of a TABLEMk entry for the damping coefficient. (Integer $\geq 0$ or blank)  |
| T(ST)      | Identification number of a TABLEMk entry for the tension stress limit. (Integer $\geq 0$ or blank)                                       |
| T(SC)      | Identification number of a TABLEMk entry for the tension compression limit. (Integer $\geq 0$ or blank)                                  |
| T(SS)      | Identification number of a TABLEMk entry for the tension shear limit. (Integer $\geq 0$ or blank)  |
| T (GEij)   | Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer > 0 or blank)        |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMk 32, G33 is modified by TABLEMk 15, and A1 is modified by TABLEMk 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT2 entry.

2. Any quantity modified by this entry must have a value on the MAT2 entry.
3. Any of the T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMk ID for the coefficient of thermal expansion is negative, the TABLEMk ordinate values will be interpreted as the thermal strain  $\varepsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative ID<sub>T(A)</sub> value will be changed to |ID<sub>T(A)</sub>| + 100000000

**MATT3****MAT3 Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT3 entry fields via TABLEMi entries that are temperature dependent.

**Format:**

| 1     | 2   | 3     | 4      | 5      | 6        | 7        | 8       | 9      | 10    |
|-------|-----|-------|--------|--------|----------|----------|---------|--------|-------|
| MATT3 | MID | T(EX) | T(ETH) | T(EZ)  | T(NUXTH) | T(NUTHZ) | T(NUZX) | T(RHO) |       |
|       |     |       |        | T(GZX) | T(AX)    | T(ATH)   | T(AZ)   |        | T(GE) |

**Example:**

|       |    |    |  |    |  |  |  |  |  |
|-------|----|----|--|----|--|--|--|--|--|
| MATT3 | 23 | 32 |  | 15 |  |  |  |  |  |
|       |    |    |  | 62 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material property identification number that matches the identification number on MAT3 entry. (Integer > 0)   |
| T(EX)      | Identification number of a TABLEMi entry for the Young's modulus in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank)                |
| T(ETH)     | Identification number of a TABLEMi entry for the thermal expansion coefficients in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank) |
| T(EZ)      | Identification number of a TABLEMi entry for the Poisson's ratio in the $x\theta$ , $\theta z$ , and $zx$ directions. (Integer $\geq 0$ or blank)       |
| T(GZX)     | Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)   |
| T(NUXTH)   | Identification number of a TABLEMi entry for the mass density. (Integer $\geq 0$ or blank)  |
| T(NUTHZ)   | Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or blank)   |
| T(NUZX)    | Identification number of a TABLEMi entry for the thermal expansion coefficients in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank) |
| T(RHO)     | Identification number of a TABLEMi entry for the thermal expansion coefficients in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank) |
| T(AX)      | Identification number of a TABLEMi entry for the Poisson's ratio in the $x\theta$ , $\theta z$ , and $zx$ directions. (Integer $\geq 0$ or blank)       |
| T(ATH)     | Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)   |
| T(AZ)      | Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)   |
| T(GE)      | Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or blank)   |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT3 entry referenced in field 2. The value recorded in a particular field of the MAT3 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, EX is modified by TABLEMi 32, EZ is modified by TABLEMi 15, and GZX is modified by TABLEMi 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT3 entry.
- Any quantity modified by this entry must have a value on the MAT3 entry.

**MATT4****Thermal Material Temperature Dependence**

Specifies table references for temperature-dependent MAT4 material properties.

**Format:**

| 1     | 2   | 3    | 4     | 5 | 6    | 7          | 8        | 9 | 10 |
|-------|-----|------|-------|---|------|------------|----------|---|----|
| MATT4 | MID | T(K) | T(CP) |   | T(H) | T( $\mu$ ) | T(HGEN ) |   |    |

**Example:**

|       |    |     |     |  |  |     |  |  |  |
|-------|----|-----|-----|--|--|-----|--|--|--|
| MATT4 | 17 | 102 | 103 |  |  | 105 |  |  |  |
|-------|----|-----|-----|--|--|-----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a MAT4 entry that is temperature dependent. (Integer $> 0$ )   |
| T(K)       | Identification number of a TABLEMj entry that gives the temperature dependence of the thermal conductivity. (Integer $\geq 0$ or blank)                       |
| T(CP)      | Identification number of a TABLEMj entry that gives the temperature dependence of the thermal heat capacity. (Integer $\geq 0$ or blank)                      |
| T(H)       | Identification number of a TABLEMj entry that gives the temperature dependence of the free convection heat transfer coefficient. (Integer $\geq 0$ or blank)  |
| T( $\mu$ ) | Identification number of a TABLEMj entry that gives the temperature dependence of the dynamic viscosity. (Integer $\geq 0$ or blank)                          |
| T(HGEN)    | Identification number of a TABLEMj entry that gives the temperature dependence of the internal heat generation property for QVOL. (Integer $\geq 0$ or blank) |

**Remarks:**

1. The basic quantities on the MAT4 entry are always multiplied by the corresponding tabular function referenced by the MATT4 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the MAT4 entry.

**MATT5****Thermal Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT5 entry fields via TABLEMi entries.

**Format:**

| 1     | 2   | 3        | 4      | 5      | 6      | 7      | 8      | 9     | 10 |
|-------|-----|----------|--------|--------|--------|--------|--------|-------|----|
| MATT5 | MID | T(KXX)   | T(KXY) | T(KXZ) | T(KYY) | T(KYZ) | T(KZZ) | T(CP) |    |
|       |     | T(HGEN ) |        |        |        |        |        |       |    |

**Example:**

|       |    |    |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|
| MATT5 | 24 | 73 |  |  |  |  |  |  |  |
|-------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a MAT5 entry that is to be temperature dependent. (Integer $> 0$ )   |
| T(Kij)     | Identification number of a TABLEMi entry. The TABLEMi entry specifies temperature dependence of the matrix term. (Integer $\geq 0$ or blank)                            |
| T(CP)      | Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity. (Integer $\geq 0$ or blank)                            |
| T(HGEN)    | Identification number of a TABLEMi entry that gives the temperature dependence of the internal heat generation property for the QVOL entry. (Integer $\geq 0$ or blank) |

**Remarks:**

1. The basic quantities on the MAT5 entry are always multiplied by the tabular function referenced by the MATT5 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the basic MAT5 entry.

**MATT8****Shell Orthotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.

**Format:**

| 1     | 2     | 3      | 4     | 5       | 6      | 7      | 8      | 9      | 10 |
|-------|-------|--------|-------|---------|--------|--------|--------|--------|----|
| MATT8 | MID   | T(E1)  | T(E2) | T(NU12) | T(G12) | T(G1Z) | T(G2Z) | T(RHO) |    |
|       | T(A1) | T(A2)  |       | T(Xt)   | T(Xc)  | T(Yt)  | T(Yc)  | T(S)   |    |
|       | T(GE) | T(F12) |       |         |        |        |        |        |    |

**Example:**

|       |    |    |  |    |  |  |  |  |  |
|-------|----|----|--|----|--|--|--|--|--|
| MATT8 | 17 | 32 |  |    |  |  |  |  |  |
|       | 15 |    |  | 52 |  |  |  |  |  |
|       |    |    |  |    |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| MAT        | Material property identification number that matches the identification number on MAT1 entry. (Integer $> 0$ )                         |
| T(E1)      | Identification number of a TABLEMi entry for the Young's modulus 1. (Integer $\geq 0$ or blank)  |
| T(E2)      | Identification number of a TABLEMi entry for the Young's modulus 2. (Integer $\geq 0$ or blank)  |
| T(NU12)    | Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer $\geq 0$ or blank)   |
| T(G12)     | Identification number of a TABLEMi entry for shear modulus 12. (Integer $\geq 0$ or blank)   |
| T(G1Z)     | Identification number of a TABLEMi entry for transverse shear modulus 1Z. (Integer $\geq 0$ or blank)                                  |
| T(G2Z)     | Identification number of a TABLEMi entry for transverse shear modulus 2Z. (Integer $\geq 0$ or blank)                                  |
| T(RHO)     | Identification number of a TABLEMi entry for mass density. (Integer $\geq 0$ or blank)   |
| T(Ai)      | Identification number of a TABLEMi entry for thermal expansion coefficient 1. See Remark 3. (Positive or negative integer, 0 or blank) |
| T(A2)      | Identification number of a TABLEMi entry for thermal expansion coefficient 2. See Remark 3. (Integer or blank)                         |
| T(Xt)      | Identification number of a TABLEMi entry for tension stress/strain limit 1. (Integer $\geq 0$ or blank)                                |
| T(Xc)      | Identification number of a TABLEMi entry for compression stress/strain limit 1. (Integer $\geq 0$ or blank)                            |

| Descriptor | Meaning   |
|------------|---|
| T(Yt)      | Identification number of a TABLEMi entry for tension stress/strain limit 2. (Integer $\geq 0$ or blank)     |
| T(Yc)      | Identification number of a TABLEMi entry for compression stress/strain limit 2. (Integer $\geq 0$ or blank) |
| T(S)       | Identification number of a TABLEMi entry for shear stress/strain limit. (Integer $\geq 0$ or blank)         |
| T(GE)      | Identification number of a TABLEMi entry for structural damping coefficient. (Integer $\geq 0$ or blank)    |
| T(F12)     | Identification number of a TABLEMi entry for Tsai-Wu interaction term. (Integer $\geq 0$ or blank)          |

**Remarks:**

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 32, A1 is modified by TABLEMi 15, and Xt is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT8 entry.
2. Any quantity modified by this entry must have a value on the MAT8 entry.
3. Any of the T(Ai) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain  $\varepsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative ID<sub>T(A)</sub> value will be changed to |ID<sub>T(A)</sub>| + 100000000.

**MATT9****Solid Element Anisotropic Material Temperature Dependence**

Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

**Format:**

| 1     | 2   | 3       | 4       | 5       | 6       | 7       | 8       | 9       | 10      |
|-------|-----|---------|---------|---------|---------|---------|---------|---------|---------|
| MATT9 | MID | T(G11)  | T(G12)  | T(G13)  | T(G14)  | T(G15)  | T(G16)  | T(G22)  |         |
|       |     | T(G23)  | T(G24)  | T(G25)  | T(G26)  | T(G33)  | T(G34)  | T(G35)  | T(G36)  |
|       |     | T(G44)  | T(G45)  | T(G46)  | T(G55)  | T(G56)  | T(G66)  | T(RHO)  | T(A1)   |
|       |     | T(A2)   | T(A3)   | T(A4)   | T(A5)   | T(A6)   |         | T(GE)   |         |
|       |     | T(GE11) | T(GE12) | T(GE13) | T(GE14) | T(GE15) | T(GE16) | T(GE22) | T(GE23) |
|       |     | T(GE24) | T(GE25) | T(GE26) | T(GE33) | T(GE34) | T(GE35) | T(GE36) | T(GE44) |
|       |     | T(GE45) | T(GE46) | T(GE55) | T(GE56) | T(GE66) |         |         |         |

**Example:**

|       |    |    |  |    |    |  |  |    |  |
|-------|----|----|--|----|----|--|--|----|--|
| MATT9 | 17 | 32 |  |    | 18 |  |  | 17 |  |
|       |    |    |  | 12 |    |  |  |    |  |
|       |    |    |  |    | 5  |  |  | 10 |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)                          |
| T(Gij)     | Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer $\geq 0$ or blank)                  |
| T(RHO)     | Identification number of a TABLEMk entry for the mass density. (Integer $\geq 0$ or blank)   |
| T(Ai)      | Identification number of a TABLEMk entry for the thermal expansion coefficients. (Integer $\geq 0$ or blank)                         |
| T(GE)      | Identification number of a TABLEMk entry for the damping coefficient. (Integer $\geq 0$ or blank)                                    |
| T(GEij)    | Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer $> 0$ or blank). |

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value recorded in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMj 32, G14 is modified by TABLEMj 18, etc. If the fields are zero or blank, then there is no temperature dependence of the fields on the MAT9 entry.
- Any quantity modified by this entry must have a value on the MAT9 entry.
- The continuation entries are optional.

## MATTEP

## Thermo-Elastic-Plastic Material Properties - SOL 400

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 400 only.

Format:

| 1      | 2          | 3  | 4     | 5      | 6 | 7   | 8   | 9    | 10 |
|--------|------------|----|-------|--------|---|-----|-----|------|----|
| MATTEP | MID        |    | T(Y0) | T(FID) |   |     |     | T(H) |    |
|        | N/A        |    |       | N/A    |   |     |     |      |    |
|        | “Chaboche” | R0 | Rinf  | B      | C | Gam | Kap | N    |    |
|        |            | Qm | $\mu$ | $\eta$ |   |     |     |      |    |

Example:

|        |     |  |    |  |  |  |  |  |  |
|--------|-----|--|----|--|--|--|--|--|--|
| MATTEP | 100 |  | 20 |  |  |  |  |  |  |
|--------|-----|--|----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MATEP entry. See Remark 1. (Integer > 0)   |
| T(Y0)      | Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)  |
| T(FID)     | Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.                                 |
| T(H)       | Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank) |
| “Chaboche” | A keyword specifying the following data pertains to the Chaboche model.   |
| R0         | Identification number of TABLEMi entry for temperature dependent R0 for isotropic hardening (Integer > 0).  |
| Rinf       | Identification number of TABLEMi entry for temperature dependent Rinfinity for isotropic hardening (Integer > 0).   |
| B          | Identification number of TABLEMi entry for temperature dependent coefficient b for isotropic hardening (Integer > 0).                                       |
| C          | Identification number of TABLEMi entry for temperature dependent coefficient C for kinematic hardening (Integer > 0).                                       |
| Kap        | Identification number of TABLEMi entry for temperature dependent Kappa value for viscosity model (Integer > 0).   |
| N          | Identification number of TABLEMi entry for temperature dependent coefficient n for viscosity model (Integer).   |
| Qm         | Identification number of TABLEMi entry for temperature dependent coefficient Qm for isotropic hardening (Integer > 0)                                       |

| Descriptor | Meaning  |
|------------|--|
| $\mu$      | Identification number of TABLEMi entry for temperature dependent coefficient $\mu$ for isotropic hardening (Integer > 0).          |
| $\eta$     | Identification number of TABLEMi entry for temperature dependent coefficient $\eta$ to introduce progressive memory (Integer > 0). |

**Remarks:**

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, SOL 400 will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

**MATTEP****Thermo-Elastic-Plastic Material Properties - SOL 600**

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

**Format:**

| 1      | 2          | 3  | 4       | 5      | 6 | 7                        | 8   | 9    | 10 |
|--------|------------|----|---------|--------|---|--------------------------|-----|------|----|
| MATTEP | MID        |    | T(Y0)   | T(FID) |   |                          |     | T(H) |    |
|        | N/A        |    | T(yc10) | N/A    |   |                          |     |      |    |
|        | "Chaboche" | R0 | Rinf    | B      | C | Gam                      | Kap | N    |    |
|        |            | Qm | $\mu$   | $\eta$ |   |                          |     |      |    |
|        | "PwrLaw"   | A  | M       | B      | N | $\sigma_0 \varepsilon_0$ |     |      |    |

**Example:**

|        |     |  |    |  |  |  |  |  |  |
|--------|-----|--|----|--|--|--|--|--|--|
| MATTEP | 100 |  | 20 |  |  |  |  |  |  |
|--------|-----|--|----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MATEP entry. See Remark 1. (Integer > 0)   |
| T(Y0)      | Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)  |
| T(FID)     | Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.                                 |
| T(H)       | Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank) |
| T(yc10)    | Identification number of TABLEMi entry for equivalent 10th cycle tensile yield stress specified in the Yc10 field of MATEP entry. (Integer > 0 or blank).   |
| "Chaboche" | A keyword specifying the following data pertains to the Chaboche model.   |
| R0         | Table for R0 for isotropic hardening. (Integer > 0)   |
| Rinf       | Table for Rinfinity for isotropic hardening (Integer > 0)   |
| B          | Table for b coefficient for isotropic hardening. (Integer > 0)  |
| C          | Table for C coefficient for kinematic hardening. (Integer > 0)  |
| Kap        | Table for Kappa value for viscosity model. (Integer > 0)  |
| N          | Table for n coefficient for viscosity model. (Integer > 0)  |
| Qm         | Table for Qm coefficient for isotropic hardening. (Integer > 0)   |
| $\mu$      | Table for $\mu$ coefficient for isotropic hardening. (Integer > 0)  |
| $\eta$     | Table for $\eta$ coefficient to introduce progressive memory. (Integer > 0)   |

| Descriptor              | Meaning  |
|-------------------------|--|
| "PwrLaw"                | A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C , ISOTROPIC option for more details. |
| A                       | Table for coefficient A. (Integer > 0)   |
| M                       | Table for coefficient m. (Integer > 0)   |
| B                       | Table for coefficient B. (Integer > 0)   |
| N                       | Table for exponent n. (Integer > 0)  |
| $\sigma_0\varepsilon_0$ | Table for $\sigma_0\varepsilon_0$ . (Integer > 0)  |

**Remarks:**

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, MSC Nastran will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

**Note:**

This Bulk Data entry accommodates Marc's input data under the model definition options TEMPERATURE EFFECTS.

**MATTF****Material Failure Model Temperature Variation**

Describes the temperature, strain rate, or other type of variation of material failure properties used in conjunction with MATF in SOL 600 and SOL 400.

**Format:**

| 1     | 2      | 3        | 4      | 5       | 6      | 7      | 8      | 9      | 10  |
|-------|--------|----------|--------|---------|--------|--------|--------|--------|-----|
| MATTF | MID    |          | T(SB)  |         |        |        |        |        |     |
|       | KIND   | Criteria | T(Xt)  | T(Xc)   | T(Yt)  | T(Yc)  | T(Zt)  | T(Zc)  | 1st |
|       | T(Sxy) | T(Syz)   | T(Szx) | T(Find) | T(Fxy) | T(Fyz) | T(Fzx) | T(Ext) |     |
|       | T(Exc) | T(Eyt)   | T(Eyc) | T(Ezt)  | T(Ezc) | T(Gxy) | T(Gyz) | T(Gzx) |     |
|       | KIND   | Criteria | T(Xt)  | T(Xc)   | T(Yt)  | T(Yc)  | T(Zt)  | T(Zc)  | 2nd |
|       | T(Sxy) | T(Syz)   | T(Szx) | F(Find) | F(Fxy) | T(Fyz) | T(Fzx) | T(Ext) |     |
|       | T(Exc) | T(Eyt)   | T(Eyc) | T(Ezt)  | T(Ezc) | T(Gxy) | T(Gyz) | T(Gzx) |     |
|       | KIND   | Criteria | T(Xt)  | T(Xc)   | T(Yt)  | T(Yc)  | T(Zt)  | T(Zc)  | 3rd |
|       | T(Sxy) | T(Syz)   | T(Szx) | T(Find) | T(Fxy) | T(Fyz) | T(Fzx) | T(Ext) |     |
|       | T(Exc) | T(Eyt)   | T(Eyc) | T(Ezt)  | T(Ezc) | T(Gxy) | T(Gyz) | T(Gzx) |     |

**Example:**

|       |     |    |    |    |    |    |    |    |     |
|-------|-----|----|----|----|----|----|----|----|-----|
| MATTF | 100 |    |    |    |    |    |    |    |     |
|       | 12  | 1  | 51 | 52 | 53 | 54 | 55 | 56 |     |
|       | 61  | 62 | 63 |    |    |    |    |    |     |
|       |     |    |    |    |    |    |    |    |     |
|       | 12  | 2  |    |    |    |    |    |    | 2nd |
|       |     |    |    |    |    |    |    |    |     |
|       | 71  | 72 | 73 | 74 | 75 | 76 | 77 | 78 |     |
|       | 12  | 4  | 81 | 82 | 83 | 84 | 85 | 86 | 3rd |
|       | 91  | 92 | 93 | 94 |    |    |    |    |     |

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a matching MATF entry. (Integer > 0; no Default; Required)   |
| KIND       | Enter the type of variation of the failure properties using the values listed below.<br>(Integer; no Default)<br>12 Temperature |

| Descriptor | Meaning   |
|------------|---|
| Criteria   | Enter same value as used by MATTF (reference only). (Integer)<br>If more than one KIND is required, enter the three lines for each KIND and Criteria as many times are required. (Integer; must be same as used by companion MATTF, reference only) |
| T(SB)      | Identification of a TABLEMi entry providing the variation of the allowable shear stress of the bonding material between layers (composites only) (Integer; Default = 0 meaning no variation)  |
| T(Xt)      | Identification of a TABLEMi entry providing the variation of the maximum tensile stress in x-direction (Integer; Default = 0 meaning no variation)  |
| T(Xc)      | Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in x-direction (Integer; Default = 0 meaning no variation)   |
| T(Yt)      | Identification of a TABLEMi entry providing the variation of the maximum tensile stress in y-direction (Integer; Default = 0 meaning no variation)  |
| T(Yc)      | Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in y-direction (Integer or blank)  |
| T(Zt)      | Identification of a TABLEMi entry providing the variation of the maximum tensile stress in z-direction (Integer; Default = 0 meaning no variation)  |
| T(Zc)      | Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in z-direction (Integer; Default = 0 meaning no variation)   |
| T(Sxy)     | Identification of a TABLEMi entry providing the variation of the maximum shear stress in xy-plane (Integer; Default = 0 meaning no variation)   |
| T(Syz)     | Identification of a TABLEMi entry providing the variation of the maximum shear stress in yz-plane (Integer; Default = 0 meaning no variation)   |
| T(Szx)     | Identification of a TABLEMi entry providing the variation of the maximum shear stress in zx-plane (Integer; Default = 0 meaning no variation)   |
| T(Find)    | Identification of a TABLEMi entry providing the variation of the Failure index (Real > 0., (Integer; Default = 0 meaning no variation)  |
| T(Fxy)     | Identification of a TABLEMi entry providing the variation of the interactive strength constant for xy-plane (Integer; Default = 0 meaning no variation)   |
| T(Fyz)     | Identification of a TABLEMi entry providing the variation of the interactive strength constant for yz-plane (Integer; Default = 0 meaning no variation)   |
| T(Fzx)     | Identification of a TABLEMi entry providing the variation of the interactive strength constant for zx-plane (Integer; Default = 0 meaning no variation)   |
| T(Ext)     | Identification of a TABLEMi entry providing the variation of the maximum tensile strain in x-direction (Integer; Default = 0 meaning no variation)  |

| Descriptor | Meaning   |
|------------|---|
| T(Exc)     | Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in x-direction (Integer; Default = 0 meaning no variation) |
| T(Yet)     | Identification of a TABLEMi entry providing the variation of the maximum tensile strain in y-direction (Integer; Default = 0 meaning no variation)                      |
| T(Eyc)     | Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in y-direction (Integer; Default = 0 meaning no variation) |
| T(Ezt)     | Identification of a TABLEMi entry providing the variation of the maximum tensile strain in z-direction (Integer; Default = 0 meaning no variation)                      |
| T(Ezc)     | Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in z-direction (Integer; Default = 0 meaning no variation) |
| T(Gxy)     | Identification of a TABLEMi entry providing the variation of the maximum shear strain in xy-plane (Integer; Default = 0 meaning no variation)                           |
| T(Gyz)     | Identification of a TABLEMi entry providing the variation of the maximum shear strain in yz-plane (Integer; Default = 0 meaning no variation)                           |
| T(Gzx)     | Identification of a TABLEMi entry providing the variation of the maximum shear strain in zx-plane (Integer; Default = 0 meaning no variation)                           |

**Remarks:**

1. See the MATTF entry for a complete description of the meaning of each of the variables for the various failure criterion.
2. If there is no variation of a particular entry, leave the field blank or enter zero.
3. Continuation entries except the last line are required. The last line is optional.

**MATTG**

## Temperature Variation of Interlaminar Materials

Specifies gasket material property temperature variation to be used in SOL 600 and SOL 400.

**Format:**

| 1     | 2     | 3     | 4     | 5    | 6    | 7    | 8     | 9     | 10 |
|-------|-------|-------|-------|------|------|------|-------|-------|----|
| MATTG | MID   | IDYM  | IDVM  | IDDM | IDLD | IDU1 | IDU2  | IDU3  |    |
|       | IDU4  | IDU5  | IDU6  | IDU7 | IDU8 | IDU9 | IDU10 | IDYPR |    |
|       | IDEPL | IDGPL | IDGAP | N/A  | N/A  | N/A  | N/A   | N/A   |    |

**Example:**

|       |      |      |    |      |      |      |  |      |  |
|-------|------|------|----|------|------|------|--|------|--|
| MATTG | 100  | 10   | 20 | 1001 | 1002 | 1003 |  |      |  |
|       |      |      |    |      |      |      |  | 1010 |  |
|       | 1020 | 1030 |    |      |      |      |  |      |  |

| Descriptor | Meaning  |
|------------|--|
| MID        | Material ID number that matches the material ID of a corresponding MATG material. (Integer > 0; Required)  |
| IDYM       | ID of TABLEMi entry that gives the temperature variation of Young's modulus for the membrane behavior of the material. (Integer $\geq 0$ or blank)   |
| IDVM       | ID of TABLEMi entry that gives the temperature variation of Poisson's ratio for the membrane behavior of the material. (Integer $\geq 0$ or blank)   |
| IDDM       | ID of TABLEMi entry that gives the temperature variation of the mass density for the membrane behavior of the material. (Integer $\geq 0$ or blank)  |
| IDLD       | ID of TABLEMi entry that gives the temperature variation of the loading curve of the material. (Integer $\geq 0$ or blank)   |
| IDUi       | ID of TABLEMi entry that gives the temperature variation of the unloading curve of the material. There can be up to 10 unloading curves and each can have a different temperature variation. If there is no unloading, there does not need to be any unloading curves. (Integer $\geq 0$ or blank) |
| IDYPR      | ID of TABLEMi entry that gives the temperature variation of the yield pressure for the out-of -plane behavior of the material. (Integer $\geq 0$ or blank)   |
| IDEPL      | ID of TABLEMi entry that gives the temperature variation of the tensile modulus for the out-of -plane behavior of the material. (Integer $\geq 0$ or blank)  |
| IDGPL      | ID of TABLEMi entry that gives the temperature variation of the transverse shear modulus for the out-of -plane behavior of the material. (Integer $\geq 0$ or blank)   |
| IDGAP      | ID of TABLEMi entry that gives the temperature variation of the initial gap for the out- -plane behavior of the material. (Integer $\geq 0$ or blank)  |

**MATTHE****Hyperelastic Material - Temperature Dependence**

Specifies temperature-dependent properties of hyperelastic (rubber-like) materials (elastomers) for nonlinear (large strain and large rotation) analysis in SOLs 106, 129, 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin model (Model=Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

| 1      | 2      | 3      | 4      | 5      | 6      | 7       | 8     | 9     | 10 |
|--------|--------|--------|--------|--------|--------|---------|-------|-------|----|
| MATTHE | MID    | Model  |        | T(K)   | T(RHO) | T(Texp) |       | T(GE) |    |
|        | T(C10) | T(C01) | T(D1)  |        |        |         |       |       |    |
|        | T(C20) | T(C11) | T(C02) | T(D2)  |        |         |       |       |    |
|        | T(C30) | T(C21) | T(C12) | T(C03) | T(D3)  |         |       |       |    |
|        | T(C40) | T(C31) | T(C22) | T(C13) | T(C04) | T(D4)   |       |       |    |
|        | T(C50) | T(C41) | T(C32) | T(C23) | T(C14) | T(C05)  | T(D5) |       |    |

Format 2: Ogden Model

| 1      | 2     | 3     | 4     | 5     | 6      | 7       | 8 | 9     | 10 |
|--------|-------|-------|-------|-------|--------|---------|---|-------|----|
| MATTHE | MID   | Model |       | T(K)  | T(RHO) | T(Texp) |   | T(GE) |    |
|        | T(D1) | T(D2) | T(D3) | T(D4) | T(D5)  |         |   |       |    |

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

| 1      | 2      | 3      | 4     | 5     | 6      | 7       | 8 | 9     | 10 |
|--------|--------|--------|-------|-------|--------|---------|---|-------|----|
| MATTHE | MID    | Model  |       | T(K)  | T(RHO) | T(Texp) |   | T(GE) |    |
|        | T(NKT) | T(N/E) | T(Im) |       |        |         |   |       |    |
|        | T(D1)  | T(D2)  | T(D3) | T(D4) | T(D5)  |         |   |       |    |

## Example - Format 1:

|        |    |        |     |     |     |     |     |  |  |
|--------|----|--------|-----|-----|-----|-----|-----|--|--|
| MATTHE | 1  | Monney |     |     |     |     |     |  |  |
|        | 10 | 1      | 100 |     |     |     |     |  |  |
|        | 20 | 11     | 2   | 200 |     |     |     |  |  |
|        | 30 | 21     | 12  | 3   | 300 |     |     |  |  |
|        | 40 | 31     | 22  | 13  | 4   | 400 |     |  |  |
|        | 50 | 41     | 32  | 23  | 14  | 5   | 500 |  |  |

## Example - Format 2:

|        |     |       |     |     |     |  |  |  |  |
|--------|-----|-------|-----|-----|-----|--|--|--|--|
| MATTHE | 1   | Ogden |     |     |     |  |  |  |  |
|        |     |       |     |     |     |  |  |  |  |
|        |     |       |     |     |     |  |  |  |  |
|        |     |       |     |     |     |  |  |  |  |
|        | 100 | 200   | 300 | 400 | 500 |  |  |  |  |

## Example - Format 3:

|        |     |        |     |     |     |  |  |  |  |
|--------|-----|--------|-----|-----|-----|--|--|--|--|
| MATTHE | 1   | Aboyce |     |     |     |  |  |  |  |
|        | 1   | 2      |     |     |     |  |  |  |  |
|        | 100 | 200    | 300 | 400 | 500 |  |  |  |  |

| Descriptor | Meaning   |  |  |  |  |  |  |  |  |
|------------|---|--|--|--|--|--|--|--|--|
| MID        | Identification number of a MATHE entry. See Remark 1. (Integer > 0; no Default)   |  |  |  |  |  |  |  |  |
| Model      | Select hyperelastic material model form. (Character; Default = Mooney)  |  |  |  |  |  |  |  |  |
|            | Mooney  | For generalized Mooney-Rivlin hyperelastic model |  |  |  |  |  |  |  |
|            | Ogden   | For Ogden hyperelastic model                     |  |  |  |  |  |  |  |
|            | Foam  | For Foam model                                   |  |  |  |  |  |  |  |
|            | Aboyce  | For Arruda-Boyce strain energy model             |  |  |  |  |  |  |  |
|            | Gent  | For Gent strain energy model                     |  |  |  |  |  |  |  |
| T(Value)   | Identification number of a TABLEMi entry for the temperature variation of the matching value from the MATHE entry - see the MATHE entry for the definition of each coefficient value. (Integer; Default = 0 which means no table variation for that particular value) |  |  |  |  |  |  |  |  |

## Remarks:

1. The MATTHE entry must have the same ID as the corresponding MATHE entry. Each table ID on the MATTHE entry corresponds to a parameter on the MATHE entry.
2. If the thermal expansion coefficient is temperature-dependent, the thermal strain is computed as

$$\varepsilon_{\text{th}} = \bar{\alpha}(T - T_{\text{ref}}) - \bar{\alpha}_0(T_0 - T_{\text{ref}})$$

where  $T_{\text{ref}}$  is the reference temperature at which measurement of the thermal expansion coefficient is based,  $T_0$  is an initial temperature, and  $\bar{\alpha}_0$  is the thermal expansion coefficient at the initial temperature  $T_0$ . If the thermal expansion is not temperature-dependent, the thermal strain expression is reduced to the usual expression:

$$\varepsilon_{\text{th}} = \bar{\alpha}(T - T_0)$$

3. If experimental data is provided, it is expected that the user has the data for multiple tests of the same type at different temperatures. The use of TABLEST's for this entry is not presently available. If experimental data is used to define the constants, use a GUI to define the constants at each temperature, enter the baseline values on the MATHE entry and the variation of each coefficient using TABLEMi.

**MATTORT**

## Thermoelastic Orthotropic Material

Specifies temperature-dependent properties of elastic orthotropic materials for linear and nonlinear analyses used in SOL 600 and SOL 400.

**Format:**

| 1       | 2      | 3      | 4      | 5     | 6       | 7       | 8       | 9      | 10 |
|---------|--------|--------|--------|-------|---------|---------|---------|--------|----|
| MATTORT | MID    | T(E1)  | T(E2)  | T(E3) | T(NU12) | T(NU23) | T(NU31) | N/A    |    |
|         | T(G12) | T(G23) | T(G31) | T(A1) | T(A2)   | T(A3)   | T(SY)   | T(WHS) |    |

**Example:**

|         |     |   |   |   |  |  |  |  |  |
|---------|-----|---|---|---|--|--|--|--|--|
| MATTORT | 100 | 5 | 6 | 7 |  |  |  |  |  |
|---------|-----|---|---|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of a MATORT entry. See Remark 1. (Integer > 0; no Default)   |
| T(Ei)      | Identification number of a TABLEMi entry for the Young's modulus as a function of temperature in each respective direction. Remark 2. (Integer > 0 or blank) |
| T(Nuij)    | Identification number of a TABLEMi entry for the Poisson's ratio as a function of temperature in each respective direction. (Integer > 0 or blank)           |
| T(Gij)     | Identification number of a TABLEMi entry for the shear modulus as a function of temperature in each respective direction. (Integer > 0 or blank)             |
| T(Ai)      | Identification number of a TABLEMi entry for the coefficient of thermal expansion as a function of temperature. (Integer > 0 or blank)                       |
| T(SY)      | Identification number of a TABLEMi entry for the yield stress as a function of temperature. (Integer > 0 or blank)   |
| T(WHS)     | Identification number of a TABLEMi entry for the work hardening slope as a function of temperature. (Integer > 0 or blank)                                   |

**Remarks:**

1. The MATTORT entry must have the same ID as the corresponding MATORT entry. Each table ID on the MATTORT entry corresponds to a parameter on the MATORT entry.
2. The table represents material constants as a function of temperature. Therefore, the curve should comprise the original value specified in the MATORT entry (most likely at room temperature).
3. For SOL 600, the continuation line is required for versions prior to MD Nastran 2010 and MSC Nastran 2009.

**MATTUSR****Specifies Table Variation of User Defined Generic Materials**

Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.

**Format:**

| 1       | 2     | 3     | 4     | 5      | 6     | 7     | 8     | 9 | 10 |
|---------|-------|-------|-------|--------|-------|-------|-------|---|----|
| MATTUSR | MID   |       |       | T(RHO) | T(A1) | T(A2) | T(A3) |   |    |
|         | T(GE) | T(ST) | T(SC) | T(SS)  |       |       |       |   |    |

**Example:**

|         |    |  |  |    |    |  |  |  |  |
|---------|----|--|--|----|----|--|--|--|--|
| MATTUSR | 20 |  |  | 51 | 52 |  |  |  |  |
|---------|----|--|--|----|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of MATUSR. (Integer > 0; no Default)   |
| T(RHO)     | Table ID of a TABLEM1 or TABL3Di for the mass density for structural analysis. (Integer, no Default)   |
| T(Ai)      | Table ID of a TABLEM1 or TABL3Di for the coefficient of thermal expansion. (Integer)<br>If T(A1) is not zero and T(A2) and T(A3) are zero, then T(A1)=T(A2)=T(A3). |
| T(GE)      | Identification number of a TABLEMk entry for the damping coefficient. (Integer $\geq 0$ or blank)  |
| T(ST)      | Identification number of a TABLEMk entry for the tension stress limit. (Integer $\geq 0$ or blank)   |
| T(SC)      | Identification number of a TABLEMk entry for the compression limit. (Integer $\geq 0$ or blank)  |
| T(SS)      | Identification number of a TABLEMk entry for the tension shear limit. (Integer $\geq 0$ or blank)  |

**MATTVE****Thermo-Visco-Elastic Material Properties**

Specifies temperature-dependent visco-elastic material properties in terms of Thermo-Rheologically Simple behavior to be used for quasi-static or transient dynamic analysis in SOL 600 and SOL 400 only.

**Format:**

| 1      | 2   | 3        | 4  | 5    | 6     | 7    | 8    | 9  | 10 |
|--------|-----|----------|----|------|-------|------|------|----|----|
| MATTVE | MID | function | RT | ENER | FRACT | TDIF | TREF | NP |    |
|        | A1  | A2       |    |      |       |      |      |    |    |
|        | C0  | C1       | C2 | C3   | C4    | etc. |      |    |    |
|        | W0  | W1       | W2 | W3   | W4    | etc. |      |    |    |
|        | T0  | T1       | T2 | T3   | T4    | etc. |      |    |    |

**Example:**

|        |      |       |      |        |      |      |      |      |       |
|--------|------|-------|------|--------|------|------|------|------|-------|
| MATTVE | 100  | WLF   | 100. |        |      |      |      |      |       |
|        | 0.8  | 1.2   |      |        |      |      |      |      |       |
| MATTVE | 101  | POWER | 100. |        |      |      |      | 11   |       |
|        | 0.0  | 0.0   |      |        |      |      |      |      |       |
|        | 1.0  | .99   | .98  | .97    | .96  | .95  | .94  | .93  | W0-7  |
|        |      | .92   | .915 | .914   |      |      |      |      | W8-10 |
| MATTVE | 102  | NARA  |      | 4.3E-8 | .75  | 200. | 400. | 11   |       |
|        | 0.0  | 0.0   |      |        |      |      |      |      |       |
|        | 1.0  | .99   | .98  | .97    | .96  | .95  | .94  | .93  | W0-7  |
|        |      | .92   | .915 | .914   |      |      |      |      | W8-10 |
|        | 235. | 234.  | 233. | 232.   | 231. | 230. | 229. | 228. | T0-7  |
|        |      | 227.  | 226. | 225.   |      |      |      |      | T8-10 |
| MATTVE | 103  | USER  | 345. |        |      |      |      | 8    |       |

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number. See Remark 1. (Integer > 0)                        |
| Function   | Name of the shift function. See Remarks 2. and 3. (Character)             |
| WLF        | Williams-Landell-Ferry form, Requires A1 and A2. (Default)                |
| POWER      | power series form   |
| NARA       | Narayanaswamy model   |
| USER       | specify the shift function with a user subroutine                         |
| RT         | Enter the reference or glass transition temperature. (Real; Default = 0.) |

| Descriptor | Meaning  |
|------------|--|
| ENER       | Used for NARA model only, enter the activation energy divided by the gas constant Q/R. (Real; no Default)  |
| FRACT      | Used for NARA only, enter the fraction parameter. (Real; no Default)   |
| TDIF       | Used for NARA only, enter the temperature shift between your temperature and absolute temperature for calculating fictitious temperatures. (Real; no Default)          |
| TREF       | Used for NARA only, enter the reference temperature for structural relaxation. (Real; no Default)  |
| NP         | For POWER, enter the number of coefficients in the power series representation. For NARA, NP is the member of terms in the Proxy series.                               |
| A1, A2     | For WLF model enter the constants A1 and A2. For other models, enter 0.0 for A1 and A2. Do not enter a blank line as it will be stripped out. (Real; no Default)       |
| Ci         | Coefficients of the shift function, enter NP values for POWER only. Do not enter Ci for NARA. (Real; no Defaults)  |
| Wi         | For NARA model only, enter the weighting factors in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults)          |
| Ti         | For the NARA model only, enter the relaxation time values in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults) |

### Remarks:

1. The MATVE Bulk Data entry with the same MID must exist for MATTVE to be effective.
2. The viscoelastic behavior is especially noticeable in the organic high polymers. There are many different kinds of such materials including various plastics, natural and synthetic rubbers. Their mechanical properties depend strongly on temperature, and these properties change drastically in the vicinity of a critical temperature called the glass-transition temperature  $T_g$ . The polymer well below  $T_g$  is an organic glass with a relatively high modulus. The viscoelastic behavior predominates in the transition range around  $T_g$ . The polymer above the transition region (but below the melting point) becomes a rubbery solid with a low modulus.

Polymers are broadly classified as amorphous polymers and polycrystalline polymers. Under stress-relaxation at a constant strain in the glass-transition region temperature, the amorphous polymer exhibits a phase change over time from the glassy state to the rubbery state. The response is manifested in the shear modulus as a function of time, in which initially high shear modulus changes into low shear modulus. The relaxation curve of the modulus in a log-log scale plot appears as a flat plateau of glassy modulus  $G_g$  shifting down to the equilibrium modulus  $G_e$  at the rubbery plateau. Such a relaxation behavior of the amorphous polymer is observed even when the temperature is well below  $T_g$  for a prolonged period of time in a very slow process. A similar behavior is found in the rubbery elastic region, but the process is faster.

Fortunately, the mechanical properties of amorphous polymers obey a time-temperature superposition principle, which allows the use of data obtained at different temperatures to extend the time scale at any given temperature. For such a behavior, the amorphous polymer is characterized as thermo-rheologically simple (TRS). Since the relaxation process extends several decades on the logarithmic time scale at lower temperatures, it is not feasible to determine the whole curve by a constant strain test at one temperature. Instead, the relaxation characteristics are measured at elevated temperatures in reduced time scale. Then the polymers exhibit a translational shift of all the characteristic responses with a change of temperature along the logarithmic time axis. This shift occurs parallel to the time axis without a change in properties: glassy and rubbery moduli. The modulus curve shifts towards shorter time with an increased temperature.

3. The reduced ( $\xi$ ), or pseudo, time is related to the actual time ( $t$ ) through a shift function which is a function of temperature, i.e.,

$$\xi(t) = \int_0^t \frac{ds}{A(T(s))}$$

where  $A(T)$  is a shift function in terms of temperature  $T$  at time  $t$ . The shift function is a material property and must be determined experimentally. A shift function approximated by Williams-Landell-Ferry, known as WLF equation, has the form:

$$\log A = h(T) = -\frac{A_1(T - T_0)}{A_2 + (T - T_0)}$$

where  $T_0$  is the reference temperature at which relaxation data are given and  $A_1, A_2$  are calibration constants obtained at this temperature. Notice that  $A = 1$  if the reduced time is the same as the actual time. If  $T \leq T_0 - A_2$ , the deformation will be elastic.

Another form of the shift function is available as a power series in  $(T - T_0)$ , i.e.,

$$\log A = h(T) = \sum_{i=0}^{10} C_i (T - T_0)^i$$

4. The WLF shift function requires A1 and A2. The power series can have a maximum 11 coefficients C0 through C10.

Note:

1. This entry matches the three options for SHIFT FUNCTION in Marc: William-Landell-Ferry, Power Series and Narayanaswamy models.
2. The user subroutine is TRSFAC in SOL 600; this is not supported in SOL 400.

## MATUDS

### Material User Defined Service or Subroutine

Allows the user to provide material routines for use with enhanced material models in SOL 400 only.

**Format:**

| 1      | 2      | 3       | 4       | 5      | 6      | 7                   | 8      | 9      | 10 |
|--------|--------|---------|---------|--------|--------|---------------------|--------|--------|----|
| MATUDS | MID    | MTYPE   | GROUP   | UNAME  |        |                     |        |        |    |
|        | “INT”  | IDATA1  | IDATA2  | IDATA3 | IDATA4 | IDATA5              | IDATA6 | IDATA7 |    |
|        |        | IDATA8  | IDATA9  | ...    | ...    | IDATA <sub>n</sub>  |        |        |    |
|        | “REAL” | RDATA1  | RDATA2  | RDATA3 | RDATA4 | RDATA5              | RDATA6 | RDATA7 |    |
|        |        | RDATA8  | RDATA9  | ...    | ...    | RDATA <sub>n</sub>  |        |        |    |
|        | “CHAR” | CADATA1 | CADATA2 | ...    | ...    | CADATA <sub>n</sub> |        |        |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE MATERIAL 'SCA.MDSolver.Util.Ums'
```

*In Bulk Data:*

|        |      |        |          |          |  |  |  |  |  |
|--------|------|--------|----------|----------|--|--|--|--|--|
| MATUDS | 17   | MATEP  | MATERIAL | UELASTOM |  |  |  |  |  |
| MATUDS | 17   | MATUSR | MATERIAL | HYPELA2  |  |  |  |  |  |
|        | REAL | .00134 | 1.467+4  | .03      |  |  |  |  |  |
|        | INT  | 8      | 3        |          |  |  |  |  |  |

**Descriptor      Meaning**

**MID**      Identification number of a MAT1, MAT2, MAT3, MAT8, MAT9, MATHE, MATHP, MATORT, MATUSR, MCOHE, or MATD\* entries.

**MTYPE**      The name of the material entry. MAT1, MATEP, MATE, MATHE, MATORT, MATUSR, MATVE, or MATVP. (Character; no Default).

**GROUP**      The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)

**UNAME**      User subroutine name associated with the entry. See Remark 6.and 7.(Character).

**“INT”**      Keyword indicating that the following data is integer. (Character)

**IDATA<sub>i</sub>**      Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)

**“REAL”**      Keyword indicating that the following data is real. (Character)

**RDATA<sub>i</sub>**      Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

- “CHAR”      Keyword indicating that the following data is Character. (Character)
- CDATAAi      Additional user supplied Character data not already existing on the specified MAT entry.  
(Character; no Default)

### Remarks:

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The MID must match an existing MID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”
5. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
6. UNAME must be truncated to 8 characters in the bulk data field.
7. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

| MTYPE  | UNAME      | SOL 400 |
|--------|------------|---------|
| MATHE  | uelastomer | X       |
| MATUSR | hypela2    | X       |
| MAT1   | crplaw     | X       |
| MATF   | ufail      | X       |
| MATF   | uprogfail  | X       |
| MATORT | orient     | X       |
| MATUSR | umat       | X       |
| MCOHE  | ucohes     | X       |

8. In SOL 400, MATUDS is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
9. The UMAT and UCOHES user subroutines support user defined state variables, see UDSESV entry.
10. For hyper-elastic(or plastic) materials. the deformation gradient is available for bricks and shells in the user-defined material subroutines, the variable IHYPER on the material card should be set to 1. The deformation gradient components  $F_{11}$ ,  $F_{21}$ ,  $F_{31}$ ,  $F_{12}$ ,  $F_{22}$ ,  $F_{32}$ ,  $F_{13}$ ,  $F_{23}$  and  $F_{33}$  can be found in the history variables array in positions IDATA2+1 to IDATA2+9, the positions coming right after the requested number of history variables. For shell elements, the components of the deformation

gradient are with respect to the co-rotational system for the element currently used. In this case the third row of the deformation gradient, the components  $F_{31}$ ,  $F_{32}$  and  $F_{33}$  will not be properly updated when entering the user-defined material routine. These components depend on the thickness strain increment which in turn must be determined so that the normal stress in the shell vanishes. For a given thickness strain increment  $d_3$ , these three components,  $f_{31}$ ,  $f_{32}$  and  $f_{33}$ , can be determined by calling the subroutine. For hyper-elastic materials there are push forward operations that can be called from within the user defined subroutines too. If the local coordinate system option is invoked (IDATA3=1), then the deformation gradient is transformed to this local system prior to entering the user-defined material routine according to:

$$\bar{F}_{ij} = Q_{ki}^s F_{kj}$$

where  $Q_{ki}^s$  refers to a transformation between the current global and material frames. For IDATA equal to 3 one can choose to put IHYPER equal to -1 which results in that the deformation gradient is transformed according to:

$$\bar{F}_{ij} = F_{ik} Q_{kj}^r$$

where  $Q_{kj}^r$  is the transformation between the reference global and material frames. For this latter option the spatial frame remains the global one so the stresses should be expressed in this frame of reference upon exiting the user defined routines. The suitable choice of IHYPER depends on the formulation of the material model. For shells, there is also the option of setting IHYPER=3 which will make the deformation gradient computed from the nodal coordinates and in the global coordinate system. With this option the user must compute the stress in the local system of interest, whence a transformation matrix between the global and this local system is passed to the user material routines. The columns in this matrix correspond to local basis vectors expressed in global coordinates, and this is the system that stress needs to be computed in. The user must be aware that since the deformation gradient is calculated directly from the element deformation it may not be consistent with the theory of the element that is used for the material. Also, the thickness used in the calculations is constant and there is currently no thickness change treatment for this option. In the following, a Neo-Hookean material is used as an example of the usage of the deformation gradient in user-defined materials. With  $\lambda$  and  $\mu$  being the Lame parameters in the linearized theory, the strain energy density for this material is given by:

$$\psi = \frac{1}{2} \lambda [\ln(\det F)]^2 - \mu \ln(\det F) + \frac{1}{2} \mu [\text{tr}(F^T F) - 3]$$

meaning that the Cauchy stress can be expressed as:

$$\sigma = \frac{1}{\det F} [\lambda \ln(\det F) I + \mu (F^T F - 1)]$$

**MATUDS****Material User Defined Service or Subroutine - SOL 600**

Allows the user to provide material routines for use with enhanced material models in SOL 600.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7                  | 8      | 9      | 10 |
|--------|--------|--------|--------|--------|--------|--------------------|--------|--------|----|
| MATUDS | MID    | MTYPE  |        | UNAME  | IOPT   |                    |        |        |    |
|        | “INT”  | IDATA1 | IDATA2 | IDATA3 | IDATA4 | IDATA5             | IDATA6 | IDATA7 |    |
|        |        | IDATA8 | IDATA9 | ...    | ...    | IDATA <sub>n</sub> |        |        |    |
|        | “REAL” | RDATA1 | RDATA2 | RDATA3 | RDATA4 | RDATA5             | RDATA6 | RDATA7 |    |
|        |        | RDATA8 | RDATA9 | ...    | ...    | RDATA <sub>n</sub> |        |        |    |
|        | “CHAR” | CDATA1 | CDATA2 | ...    | ...    | CDATA <sub>n</sub> |        |        |    |

**Examples:**

|        |      |        |        |            |      |    |        |  |  |
|--------|------|--------|--------|------------|------|----|--------|--|--|
| MATUDS | 17   | MATEP  |        | uelastomer | 2    |    |        |  |  |
| MATUDS | 33   | MATHYP |        | hypela2    | 13   |    |        |  |  |
|        | REAL | 1.0E-4 | 2.0E-6 | 4.8E-6     | 38.6 |    | 1.2E-4 |  |  |
|        | INT  | 1      | 3      | 51         | 52   | 53 | 54     |  |  |
|        |      | 55     | 56     |            |      |    |        |  |  |

| Descriptor         | Meaning   |
|--------------------|---|
| MID                | Material property identification number that matches the identification number on a MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, or MATHE, primary material entry or a MATVE, MATVP, MATEP, MATF, associated material entry for structural analysis. MAT4 and MAT5 material entries for heat analysis. COHESIV for Cohesive zone modeling. (Integer > 0) |
| MTYPE              | The name of the material entry. MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHE, MATG, MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, MAT4, MAT5, COHESIV. (Character; no Default)   |
| UNAME              | A primary or secondary name to identify the user subroutine. See Remark 6. (Character; no Default)  |
| IOPT               | A user subroutine option flag. See Remark 9. (Character; Default blank)   |
| “INT”              | Keyword indicating that the following data is integer. (Character)  |
| IDATA <sub>i</sub> | Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)  |
| “REAL”             | Keyword indicating that the following data is real. (Character)   |
| RDATA <sub>i</sub> | Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)  |

|        |   |
|--------|---|
| “CHAR” | Keyword indicating that the following data is Character. (Character)  |
| CDATAi | Additional user supplied Character data not already existing on the specified MAT entry.<br>(Character; no Default) |

**Remarks:**

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials.
2. If the matching material entry is not present, the RDATA and IDATA fields may be used to construct the applicable material entry in which case the real and integer field values are in the order shown for the material entry (MATxxx and MATTxxx). The applicable material entries will be internally constructed from the MID, RDATA and IDATA fields.
3. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
4. This entry triggers a call to a user subroutine.
5. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

| MTYPE | EVAL       | SOL 600 |
|-------|------------|---------|
| MATHE | uelastomer | X       |
| MATVP | crplaw     | X       |
|       | vswell     | X       |
|       | crpvis     | X       |
|       | ucrplw     | X       |
|       | uvscpl     | X       |

Material-related structural and heat transfer user subroutines documented in Marc Volume D “User Subroutines and Special Routines” may be entered using MATUDS in addition to those tabulated above. They may alternatively be entered using Bulk Data entry [USRSUB6, 3305](#).

6. If IDATAi values are defined they are stored in common block /userii-mid/ ival(1000) Up to 1000 integer values may be specified. The userii common block would then be placed in the user subroutine along with other coding.
7. If RDATAi values are defined they are stored in common block /userr8-mid/ rvals(1000) Up to 1000 real\*8 values may be specified. The userr8 common block would then be placed in the user subroutine along with other coding.
8. The MATUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. When USERSUB is used, flags such as IUSER on the MATVP entry or parameters that are needed will be filled in automatically although they may also be entered if desired. EVAL will be stored as a character\*16 name in common block /userch/
9. This name identifies the user subroutine name to be called.

10. IOPT is a user subroutine option flag required by certain MATxxx entries. See the applicable MATxxx entry for values IOPT may use.

**MATUSR****Defines Generic Material Properties for Hypoelastic or User Material Model**

Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only. This entry is used in conjunction with the MATUDS option to activate user subroutine md\_hypela2 (SOL 400) or hypela2 (SOL 600) for planar, shell or solid elements. For nonintegrated beam elements it is used in conjunction with user subroutine ubeam (SOL 600).

**Format:**

| 1      | 2   | 3     | 4      | 5   | 6  | 7  | 8  | 9    | 10 |
|--------|-----|-------|--------|-----|----|----|----|------|----|
| MATUSR | MID | IPREF | IKINEM | RHO | A1 | A2 | A3 | TREF |    |
|        | GE  | ST    | SC     | SS  |    |    |    |      |    |

**Example:**

|        |    |   |   |        |        |  |  |  |  |
|--------|----|---|---|--------|--------|--|--|--|--|
| MATUSR | 20 | 1 | 2 | 1.0E-4 | 2.0E-6 |  |  |  |  |
|--------|----|---|---|--------|--------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number. (Integer > 0; no Default)   |
| IPREF      | Flag to indicate that user defined material is given with respect to user defined coordinate system. (Integer; Default = 0; see Remark 14.)<br>0 User stress-strain law is isotropic (Default)<br>1 User stress strain is with respect to user defined coordinate system   |
| IKINEM     | Flag to indicate which additional kinematic quantity is passed into user subroutine. Incremental and total strains are always passed into user subroutine. (Integer; Default = 0 , see Remark 3.)<br>0 Pass in incremental strain $\Delta\epsilon$ only..<br>1 Pass in $\Delta\epsilon$ , deformation gradient (F) and rotation (R)<br>2 Pass in $\Delta\epsilon$ , deformation gradient (F) and stretch ratios ( $\lambda$ )<br>3 Pass in $\Delta\epsilon$ , F, R and $\lambda$<br>13 Pass in $\Delta\epsilon$ F, R and $\lambda$ . Kinematic quantities are calculated at the mid increment if large strain is requested resulting in logarithmic strains.<br>23 Pass in $\Delta\epsilon$ F, R and $\lambda$ . Kinematic quantities are calculated at the end of the increment if large strain is requested. |
| RHO        | Mass density for structural analysis (Real $\geq$ 0; no Default)   |
| Ai         | Coefficient of thermal expansion. (Real; no Default)   |
| TREF       | Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal coefficient. See Remarks 6., 8. and 15. (Real; Default = 0.0)   |

| Descriptor | Meaning  |
|------------|--|
| GE         | Structural element damping coefficient. See Remarks 7., 8., 9. and 15.   |
| ST,SC,SS   | Stress limits for tension, compression and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. See Remark 15. (Real; Default = 0.0) |

**Remarks:**

1. This option is used together with MATUDS to define a generic material model. MATUDS is not required for SOL 600.
2. Only enter those fields necessary for the analysis being performed.
3. For SOL 400, IKINEM should be greater than 0 and user subroutine HYPELA2 is used. For SOL 600, if IKINEM=0, then subroutine HYPELA is used, else HYPELA2.
4. The generic material model is able to represent a nonlinear material behavior. For this constitutive theory, it is assumed that

$$\dot{\sigma}_{ij} = L_{ijkl} \dot{\varepsilon}_{kl} + g_{ij}$$

where  $L$  is a function of the mechanical strain and  $g$  is a function of the temperature.

The stress and strains are true stresses and logarithmic strains, respectively, when used in conjunction with large strain requests.

When used in conjunction with PARAM,LGDISP,1 (without large strain) the above equation is expressed as

$$\dot{S}_{ij} = L_{ijkl} \dot{E}_{kl} + g_{ij}$$

where  $E$  and  $S$  are the Green-Lagrange strain and second Piola-Kirchhoff stress, respectively.

5. For the MD\_HYPELA2 user subroutine, in order to provide an accurate solution, a tangent stiffness should be evaluated at each iteration. In addition, the total stress should be defined as its exact value at the end of the increment. This allows the residual load correction to work effectively. In MD\_HYPELA2, additional information is available regarding the kinematics of deformation. In particular, the deformation gradient ( $F$ ), rotation tensor ( $R$ ), and the eigenvalues ( $\lambda$ ) and eigenvectors ( $N$ ) to form the stretch tensor ( $U$ ) are also provided. This information is available only for the continuum elements namely: plane strain, generalized plane strain (SOL 600 only), plane stress, axisymmetric, axisymmetric with twist, and three-dimensional cases.
6. TREF is used as the reference temperature for the calculation of the thermal loads. TEMPERATURE (INITIAL) may be used for this purpose, in which case TREF should be blank
7. Note that the structural damping will change with time, to obtain the damping coefficient, multiply the critical damping ratio  $C/C_0$  by 2.
8. TREF and GE are ignored if this entry is referenced by a PCOMP or a PCOMPG entry.
9. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See [Parameter Descriptions, 784](#).

10. This material model should not be used with the PSHELL option or with PBARN1, PBEAMN1 or PCOMP(G) and smeared section integration.
11. For SOL 400 the user subroutines md\_hypela2 or md\_ubeam are used. For SOL 600 user subroutine hypela or hypela2 and ubeam are used, unless a MATUDS option is also included.
12. For SOL 400 this option automatically invokes the PSHLN1, PSHLN2, PSLDN1 element formulation
13. In a thermal-mechanically coupled simulation, MAT4 or MAT5 should be used to define the material behavior.
14. For SOL 600, IPREF = 0 means user subroutines ANEXP and ORIENT are not called. IPREF = 1 means the two routines are called.
15. TREF, GE, ST, SC and SS are ignored in SOL 600.
16. If A1 is given and A2, A3 are not then isotropic thermal expansion and A1 = A2 = A3.
17. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**MATVE**

## Visco-Elastic Material Properties

Specifies visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 600 and SOL 400.

Format for Model = Iso:

| 1     | 2   | 3     | 4      | 5      | 6  | 7   | 8  | 9   | 10 |
|-------|-----|-------|--------|--------|----|-----|----|-----|----|
| MATVE | MID | Model | Alphas | Alpha1 | G1 | Td1 | K1 | Tv1 |    |
|       | G2  | Td2   | G3     | Td3    | G4 | Td4 | G5 | Td5 |    |
|       | K2  | Tv2   | K3     | Tv3    | K4 | Tv4 | K5 | Tv5 |    |

Format for Model = Iso1:

| 1     | 2   | 3     | 4      | 5      | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-------|--------|--------|---|---|---|---|----|
| MATVE | MID | Model | Alphas | Alphal |   |   |   |   |    |
|       | G1  | Td1   | K1     | Tv1    |   |   |   |   |    |
|       | G2  | Td2   | K2     | Tv2    |   |   |   |   |    |
|       | G3  | Td3   | K3     | Tv3    |   |   |   |   |    |
|       | G4  | Td4   | K4     | Tv4    |   |   |   |   |    |
|       | G5  | Td5   | K5     | Tv5    |   |   |   |   |    |
|       | Gi  | Tdi   | Ki     | Tvi    |   |   |   |   |    |

Format for Model = Mooney, Ogden and Foam:

| 1     | 2   | 3     | 4      | 5      | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-------|--------|--------|-----|-----|-----|-----|----|
| MATVE | MID | Model | Alphas | Alphal | Wd1 | Td1 | Wv1 | Tv1 |    |
|       | Wd2 | Td2   | Wd3    | Td3    | Wd4 | Td4 | Wd5 | Td5 |    |
|       | Wv2 | Tv2   | Wv3    | Tv3    | Wv4 | Tv4 | Wv5 | Tv5 |    |

Format for Model = Mooney1, Ogden1 and Foam1:

| 1     | 2   | 3     | 4      | 5      | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-------|--------|--------|---|---|---|---|----|
| MATVE | MID | Model | Alphas | Alphal |   |   |   |   |    |
|       | Wd1 | Td1   | Wv1    | Tv1    |   |   |   |   |    |
|       | Wd2 | Td2   | Wv2    | Tv2    |   |   |   |   |    |
|       | Wd3 | Td3   | Wv3    | Tv3    |   |   |   |   |    |
|       | Wd4 | Td4   | Wv4    | Tv4    |   |   |   |   |    |
|       | Wd5 | Td5   | Wv5    | Tv5    |   |   |   |   |    |
|       | Wdi | Tdi   | Wvi    | Tvi    |   |   |   |   |    |

Format for ORTHO:

| 1     | 2    | 3     | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|-------|------|-------|------|------|------|------|------|---|----|
| MATVE | MID  | ORTHO |      |      |      |      |      |   |    |
|       | Td1  | Exx1  | Eyy1 | Ezz1 | Vxy1 | Vyz1 | Vzx1 |   |    |
|       | Gxy1 | Gyz1  | Gzx1 |      |      |      |      |   |    |
|       | Td2  | Exx2  | Eyy2 | Ezz2 | Vxy2 | Vyz2 | Vzx2 |   |    |
|       | Gxy2 | Gyz2  | Gzx2 |      |      |      |      |   |    |
|       | Td3  | Exx3  | Eyy3 | Ezz3 | Vxy3 | Vyz3 | Vzx3 |   |    |
|       | Gxy3 | Gyz3  | Gzx3 |      |      |      |      |   |    |
|       | Td4  | Exx4  | Eyy4 | Ezz4 | Vxy4 | Vyz4 | Vzx4 |   |    |
|       | Gxy4 | Gyz4  | Gzx4 |      |      |      |      |   |    |
|       | Td5  | Exx5  | Eyy5 | Ezz5 | Vxy5 | Vyz5 | Vzx5 |   |    |
|       | Gxy5 | Gyz5  | Gzx5 |      |      |      |      |   |    |
|       | Tdi  | Exxi  | Eyyi | Ezzi | Vxyi | Vysi | Vzxi |   |    |
|       | Gxyi | Gyzi  | Gzxi |      |      |      |      |   |    |

Example

| 1     | 2 | 3   | 4 | 5 | 6    | 7    | 8 | 9 | 10 |
|-------|---|-----|---|---|------|------|---|---|----|
| MATVE | 1 | ISO |   |   | 3030 | 0.98 |   |   |    |
|       |   |     |   |   |      |      |   |   |    |
|       |   |     |   |   |      |      |   |   |    |

| 1     | 2    | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|------|------|---|---|---|---|---|---|----|
| MATVE | 1    | ISO1 |   |   |   |   |   |   |    |
|       | 3030 | 0.98 |   |   |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| MID        | Identification number of primary material (MAT1 or MAT3 or MAT8 or MATROT or MATHE) entry. (Integer > 0). See Remark 2.  |
| Model      | Selects a visco-elastic model defining time-dependent deformation behavior (Character): ISO1 for isotropic materials referenced by MAT1 (Default) (an alternate name is Linear). ORTHO if referenced by MAT3, MAT8 and MATORT. MOONEY1 for Mooney-Rivlin model if referenced by MATHE. OGDEN1 for Ogden model if referenced by MATHE. FOAM1 for foam model if referenced by MATHE (SOL 400 only). See Remarks 2, 3, 4, 6, and 8. |
| Alphas     | Solid coefficient of thermal expansion (Real; Default = 0). See Remark 6.  |
| Alphal     | Liquid coefficient of thermal expansion (Real; Default = 0). See Remark 6.   |
| Gi         | Shear modulus for $i^{\text{th}}$ term in Prony series (Real > 0.; Default = 0).   |
| Ki         | Bulk modulus for $i^{\text{th}}$ term in Prony series (Real > 0.; Default = 0).  |
| Wdi        | Multiplier (scale factor) for $i^{\text{th}}$ term deviatoric behavior in Prony series (Real $\geq$ 0.; Default = 0). for Hyperelastic Material.   |
| Wvi        | Multiplier (scale factor) for $i^{\text{th}}$ term volumetric behavior in Prony series (Real $\geq$ 0.; Default = 0.). See Remark 7.   |
| Tdi        | Defines time constants for deviatoric behavior in Prony series (Real $\geq$ 0.; Default = 0). See Remark 7.  |
| Tvi        | Defines time constants for volumetric behavior in Prony series (Real $\geq$ 0.; Default = 0). See Remark 7.  |

### Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. The time-dependent behavior in the viscoelastic material is modeled by a Prony series expression for both small and large strain problems. The stress relaxation behavior can be modeled by relaxation functions of the shear modulus and bulk modulus and total strain energy in terms of a series of exponential decay terms, which is known as the Prony series. This is equivalent to the generalized Maxwell model, which consists of many Maxwell models connected in parallel along with an elastic spring representing a long-term behavior. The constitutive behavior of viscoelasticity depends not only on the current state of stress and strain, but also on the entire history of the development of these states. Such a behavior is most readily expressed by hereditary or Duhamel integral. The Prony series is a discrete form of this hereditary integral.
3. ISO1 supports both volumetric and deviatoric viscoelastic behavior by allowing deviatoric terms (Gi and Tdi) and volumetric terms (Ki and Tvi) in the Prony series. The shear and bulk moduli for isotropic material (ISO1) can be expressed in a Prony series as

$$G(t) = G^\infty + \sum_{i=1} G_i e^{-t/T_{di}}$$

$$K(t) = K^\infty + \sum_{i=1} K_i e^{-t/T_{di}}$$

where instantaneous values are given by:

$$G^0 = G^\infty + \sum_{i=1} G_i$$

$$K^0 = K^\infty + \sum_{i=1} K_i$$

In terms of instantaneous values, the shear and the bulk modulus relaxation can also be written as.

$$G(t) = G^0 - \sum_{i=1} G_i (1 - e^{-t/T_{di}})$$

$$K(t) = K^0 - \sum_{i=1} K_i (1 - e^{-t/T_{di}})$$

If a Prony series is selected, at least one set of modulus and time constant must be provided i.e., (G1, Td1) and /or (K1, Tv1).

4. ORTHO supports only deviatoric viscoelastic behavior by allowing deviatoric terms ( $G_{ij}$  and  $T_{di}$ ) in the Prony series. For orthotropic material behavior, the relaxation coefficients are applied to the shear modulus as

$$G_{ij}(t) = G_{ij}^0 - \sum_{n=1} G_{ij}^n (1 - e^{-t/T_{di}})$$

where  $G_{ij}^n$  is the shear modulus for  $n^{\text{th}}$  term is Prony series (Real > 0; Default = 0) for Orthotropic Material. (Here  $G_{ij}$  imply  $G_{xy}$ ,  $G_{yz}$  and  $G_{zx}$  as mentioned in ORTHO table).

5. In case of a viscous hyperelastic material, total strain energy be expressed as a Prony series expansion in terms of energy functional

$$\psi = \psi^\infty + \sum_{n=1} \psi_n e^{-t/T_{di}}$$

Above equation can also be expressed as a function of the elastic strain energy density for instantaneous deformations  $\psi^\circ$ .

$$\psi = \psi^\circ + \sum_{n=1} W_n \psi^0 e^{-t/T_i}$$

where  $W_i$  are weighting factors  $\psi_i/\psi^0$  and  $T_i$  are time constants. Two Prony series can be formed based on deviatoric ( $\psi^d$ ) and volumetric strain energy ( $\psi^v$ ). If a viscous hyperelastic Prony series is selected, at least one pair of weighting factor and time constant must be provided: (Wd1, Td1) and /or (Wv1, Tv1). OGDEN support both deviatoric (Wdi, Tdi) and volumetric strain energy (Wvi, Tvi) Prony series. However, Mooney supports only deviatoric strain energy (Wdi, Tdi) Prony series. Foam is defined in total strain energy density function and specified through  $W_i$  and  $T_i$ .

6. If ALPHAS or ALPHAL is specified, the thermal expansion coefficient specified in MAT1, MATT1, MATHE, MATTHE, MATORT or MATTORT will be ignored. Use of ALPHAS or ALPHAL requires the MATTVE Narayanaswami model.
7. If the weighting factor is left blank, the relaxation function corresponding to that coefficient is omitted.
8. In SOL 400, MATVE is used only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the MATVE data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS, SPROPMAP, -1 in the bulk data.
9. FOAM (FOAM1) no difference between deviatoric and volumetric multipliers and relaxation time. Program will use deviatoric input to apply on entire strain energy function.

#### Notes:

1. The Prony series uses values of  $G_i$  or  $K_i$  for Isotropic and Orthotropic materials. It uses weighting ( $W_i$ ) functions for Mooney, Ogden and Foam materials.
2. The instantaneous material properties are specified on primary material entries (i.e., for MAT1 or MAT3 or MAT8 or MATORT or MATHE)
3. In the MATVE entries, material properties with the Prony series parameters are specified.
4. MATVE with "ISO", "MOONEY", "OGDEN", "FOAM" models are similar to the corresponding "ISO1", "MOONEY1", "OGDEN1", "FOAM1" models with the only exception being that the former models are limited to 5 terms in the Prony series. With the latter models, there is no limit to the number of terms in the Prony series.

**MATVE**

## Visco-Elastic Material Properties- SOL700

Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 700.  
Used in SOL 700 only.

Format (for types Iso, Mooney, Ogden and Foam):

| 1     | 2   | 3     | 4 | 5 | 6  | 7  | 8    | 9    | 10 |
|-------|-----|-------|---|---|----|----|------|------|----|
| MATVE | MID | Model |   |   | G0 | GI | BETA | VISC |    |

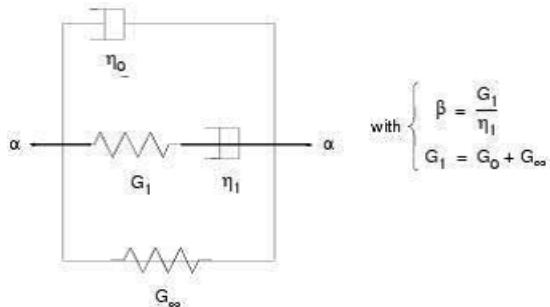
Example:

|       |   |        |  |  |      |      |     |  |  |
|-------|---|--------|--|--|------|------|-----|--|--|
| MATVE | 1 | ISOLIN |  |  | 8.E7 | 1.E7 | 0.1 |  |  |
|-------|---|--------|--|--|------|------|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MAT1 or MATHE entry (Integer > 0; required).   |
| Model      | Selects a visco-elastic model defining time-dependent deformation behavior (Character):<br>ISOLIN for linear isotropic materials referenced by MAT1 |
| G0         | Short-time shear modulus. (Real; default=0.0)   |
| GI         | Long-time shear modulus. (Real; default=0.0)  |
| BETA       | Decay constant. (Real; default=0.0)   |
| VISC       | Shear viscosity constant. (Real; default=0.0)   |

Remarks:

1. The spring-damper analog of this model is:



2. The deviatoric stress is given by

$$\sigma'_{ij}(t) = 2G_1\varepsilon'_{ij}(t) + 2 \int_0^t G(t-\tau) + \frac{\partial \varepsilon'_{ij}(\tau)}{\partial \tau} d\tau + 2\eta_0 \frac{\partial \varepsilon'_{ij}(t)}{\partial t}$$

$$G(t - \tau) = G(G_0 - G_I)e^{-\beta(t - \tau)}$$

The above equation for the deviatoric stress is the integral form of the differential equation

$$\dot{\sigma}'_{ij} + \beta\sigma'_{ij}t = 2\eta_0\ddot{\varepsilon}'_{ij} + (2\eta_0\beta + 2G_0)\dot{\varepsilon}'_{ij} + 2G_I\beta\varepsilon'_{ij}$$

A special case is for which is often written  $\eta_0 = G_I = 0$

$$\dot{\varepsilon}'_{ij} = \dot{\varepsilon}'_{ij\text{elastic}} + \dot{\varepsilon}'_{ij\text{viscous}} = \frac{\dot{\sigma}'_{ij}}{2G_0} + \frac{\beta}{2G_0}\sigma'_{ij}$$

**MATVP****Viscoplastic or Creep Material Properties**

Specifies viscoplastic or creep material properties to be used for quasi-static analysis in SOL 600 and SOL 400 only.

Primary Format: (for SOL 600, also enter Bulk Data parameter, PARAM,MARCMATT,1)

| 1     | 2   | 3 | 4    | 5 | 6 | 7 | 8 | 9     | 10 |
|-------|-----|---|------|---|---|---|---|-------|----|
| MATVP | MID | A | IT3D | M | N | P | Q | IUSER |    |

Primary Format: (for SOL 400 only, using ANAND solder creep model)

| 1     | 2    | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|-------|------|-------|-------|-------|-------|-------|-------|-------|----|
| MATVP | MID  | ANAND | PREXF | ACTEN | MULST | STNRT | SATCO | STNSA |    |
|       | HRCN | STNHR | DEFRS |       |       |       |       |       |    |

Alternate Format: (SOL 600 only, leave out PARAM,MARCMATT or enter PARAM,MARCMATT,-1))

| 1     | 2   | 3    | 4     | 5      | 6      | 7    | 8    | 9     | 10 |
|-------|-----|------|-------|--------|--------|------|------|-------|----|
| MATVP | MID | Form | Coeff | Stress | Strain | Temp | Time | IUSER |    |

Example 1: (Primary Format, A is built into TABL3Di with IT3D=20)

|       |    |     |    |     |     |     |     |  |  |
|-------|----|-----|----|-----|-----|-----|-----|--|--|
| MATVP | 10 | 1.0 | 20 | 1.5 | 1.1 | 1.0 | 1.0 |  |  |
|-------|----|-----|----|-----|-----|-----|-----|--|--|

Example 2: (Primary Format, A is provided along with exponents for stress, strain, temperature and time)

|       |    |        |   |     |     |     |     |  |  |
|-------|----|--------|---|-----|-----|-----|-----|--|--|
| MATVP | 10 | .00375 | 0 | 1.5 | 1.1 | 1.0 | 1.0 |  |  |
|-------|----|--------|---|-----|-----|-----|-----|--|--|

Example 3: (Primary Format, user subroutines crplaw.f and vswell.f are used)

|       |    |     |  |  |  |  |  |  |  |
|-------|----|-----|--|--|--|--|--|--|--|
| MATVP | 10 | 0.0 |  |  |  |  |  |  |  |
|-------|----|-----|--|--|--|--|--|--|--|

Example 4: (Primary Format, ANAND solder creep model is used)

|       |         |       |        |       |   |       |       |         |  |
|-------|---------|-------|--------|-------|---|-------|-------|---------|--|
| MATVP | 2       | ANAND | 2.23e4 | 15583 | 7 | 0.143 | 72.73 | 0.00437 |  |
|       | 1787.02 | 3.73  | 15.09  |       |   |       |       |         |  |

Example 5: (Alternate Format with 4 table inputs, one each for stress, strain, temperature and time)

|       |     |       |         |     |     |     |     |  |  |
|-------|-----|-------|---------|-----|-----|-----|-----|--|--|
| MATVP | 100 | TABLE | 3.5E-15 | 101 | 102 | 103 | 104 |  |  |
|-------|-----|-------|---------|-----|-----|-----|-----|--|--|

Example 6: (Alternate Format, user subroutine ucrplw.f is used)

|       |     |      |  |  |  |  |  |  |  |
|-------|-----|------|--|--|--|--|--|--|--|
| MATVP | 100 | USER |  |  |  |  |  |  |  |
|-------|-----|------|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of MAT1, MAT2, MATORT or MAT9 entry. See Remark 1. (Integer > 0)  |
| A          | Enter the coefficient A in for the equations of Remark 2. (this value could alternatively be built into the table referenced using IT3D in which case A should be set to 1.0. If A=0.0, user subroutine(s) designated by the flag in field 9 may be used to define the creep law and fields 4-8 will be ignored if entered. (Real; no Default)  |
| IT3D       | ID of a TABL3Di entry that defines the variation with respect to stress, strain, temperature and/or time per the equations discussed in Remark 2. The exponents are provided in the next 4 fields. (Integer > 0; Default = 0)   |
| M          | Exponent m for stress in equation in Remark 2. (Real; Default = 0.0)  |
| N          | Exponent n for strain in equation in Remark 2. (Real; Default = 0.0)  |
| P          | Exponent p for temperature in equation in Remark 2. (Real; Default = 0.0)   |
| Q          | Exponent q for time in equation in Remark 2. (Real; Default = 0.0)  |
| IUSER      | Packed list designating which user subroutines (if any) apply to this analysis. The user must confirm that the selected user subroutines are appropriate for the current analysis and are consistent with other entries in the model. (Integer; Default = 0) <ul style="list-style-type: none"> <li>1 crplaw.f</li> <li>2 vswell.f (only available in SOL 600)</li> <li>3 crpvis.f (only available in SOL 600)</li> <li>4 ucrplw.f (only available in SOL 600)</li> <li>5 uvscpl.f (only available in SOL 600)</li> </ul> (examples, if crplaw.f is used, enter 1 if both crplaw.f and vswell.f are required, enter 12) |
| Form       | (Alternate format only) Selects a creep data input form defining creep strain rate from the options listed below. (Character; no Default): <ul style="list-style-type: none"> <li>"POWER" for exponent input in power law form</li> <li>"TABLE" for piece-wise linear curve input in TABLEM1 entry.</li> </ul>  |

| Descriptor | Meaning  |
|------------|--|
| "USER"     | user subroutine crplaw.f will be used instead of the values/table ID's on the entry (if this form is used, fields 4-9 will be ignored if entered).   |
|            | See Remark 2.  |
| Coeff      | Specifies the coefficient value, A in equation. (Real > 0)   |
| Stress     | Identification number of TABLEM1 (Integer > 0) for the function f or exponent m (Real > 0.) for an effective stress function, depending on the Form field.   |
| Strain     | Identification number of TABLEM1 (Integer > 0) for the function g or exponent n (Real; Default = 0.) for an equivalent creep strain function, depending on the Form field.   |
| Temp       | Identification number of TABLEM1 (Integer > 0) for the function h or exponent p (Real; Default = 0) for a temperature function, depending on the Form field.   |
| Time       | Identification number of TABLEM1 (Integer > 0) for the function K or exponent q (Real > 0; Default = 1) for a time function, depending on the Form field.  |
| ANAND      | A keyword specifying the Anand solder creep model. The Anand solder creep model consists of a simple set of constitutive equations for large, isotropic, viscoplastic problems. If this key word is used, next 9 fields for parameters of Material properties must be filled with values. Please note that Anand solder creep model is only for isotropic materials. See Remark 12. and 13.. |
| PREXF      | Pre-exponential factor. $A, (s^{-1})$  |
| ACTEN      | Activation Energy. $Q$   |
| MULST      | Multiplier of stress. $?$  |
| STNRT      | Strain rate sensitivity of stress. $M$   |
| SATCO      | Deformation resistance saturation coefficient. $\hat{s}, (\text{MPa})$   |
| STNSA      | Strain rate sensitivity of saturation. $n$   |
| HRCN       | Hardening constant. $h_0, (\text{MPa})$  |
| STNHR      | Stain rate sensitivity of hardening. $A$   |
| DEFRS      | Initial value of deformation resistance.<br>$s_0, (\text{MPa})$  |

### Remarks:

1. This Bulk Data entry is activated if a MAT1, MAT2, MATORT, or MAT9 entry with the same MID is specified in an implicit nonlinear analysis. This creep capability is available for isotropic, orthotropic, and anisotropic elasticity, which can be coupled with plasticity using MATEP entry. Coupling with plasticity is allowed only for selected plasticity models, which include von Mises, Hill's anisotropy (creep stays isotropic), and Mohr-Coulomb models. However, viscoelasticity (MATVE) cannot be combined with viscoplasticity (MATVP).

Please note that ANAND solder creep model may be associated with isotropic material, i.e., MAT1, only.

2. The creep behavior of the material is expressed in terms of creep strain rate as a product of a number of terms (functions of effective stress, equivalent creep strain, temperature, and time) in either piece-wise linear curves or exponential form, i.e.,  $\dot{\varepsilon}^c = A \cdot \bar{\sigma}^m \cdot (\bar{\varepsilon}^c)^n \cdot T^p \cdot (qt^{q-1})$

$$\text{or } \dot{\varepsilon}^c = A \cdot f(\bar{\sigma}) \cdot g(\bar{\varepsilon}^c) \cdot h(T) \cdot \frac{dK(t)}{dt}$$

Power-Law creep (exponential form) can be used with either explicit or implicit creep methods. The choice of explicit or implicit creep is made via the NLMOPTS option.

- a. Explicit creep can be combined with other plasticity options defined through the MATEP card. The supported plasticity models include Von Mises, Hill's Anisotropy (creep remains isotropic) and Mohr-Coulomb models. The creep and plastic strains are treated separately with the algorithm handling the creep in an explicit manner and handling the plasticity in an implicit manner.
- b. Implicit power law creep can be combined with the Von Mises yield criterion defined through the MATEP card. Both the creep and plasticity are handled simultaneously through an implicit viscoplasticity algorithm and a combined inelastic strain is computed.
- c. Note that for the explicit creep strain-rate evaluation, N should only be specified if some creep strain is already present in the model. Otherwise, N should be zero (blank), or implicit creep should be selected in CREEP of NLMOPTS.
- d. The dependence of time (Q) is specified as function of total equivalent creep strain i.e.,  $e_c = A t^Q$  (i.e.,  $\text{dot}(e_c) = A Q t^{Q-1}$ ).

The functions f, g, h and K are specified as piece-wise linear functions in a tabular form using TABLEM1 entry, if the Table Form is selected. Notice that the last term in time shows function K for the equivalent creep strain in terms of time, instead of creep strain rate.

The creep strain from the creep material is a permanent strain unlike the creep strain for materials using the CREEP Bulk Data entry. As such, this creep material may be classified as viscoplastic material. This creep capability is provided for the primary and the secondary creep behavior, because the tertiary creep involves material instability such as necking.

3. For SOL 600, see associated Bulk Data entry, MPCREEP.
4. For SOL 400 see associated Bulk Data entry, NLMOPTS.
5. The alternate format is determined by field 3 with a character string POWER, TABLE, or USER. The alternative format is currently only available for SOL 600 not SOL 400.
6. There must be a non-blank entry in field 3 for either format.
7. In SOL 400, MATVP can be used with NLPARM only when total time (NINC \* DT = 1.0). It is recommended that MATVP be used in conjunction with NLSTEP.
8. Furthermore, for NLSTEP adaptive stepping, use of TABSCTL to specify user criteria on creep strains and stresses is also recommended.

9. For SOL 600, MATVP is used in conjunction with NLPARM with no restrictions on the value of the total time (NINC \* DT). NLAUTO may be added to specify under stress, strain or other criteria and/or for advanced convergence controls not available on the first line of NLPARM. Additional convergence and solver controls are available using NLSTRAT.
10. In SOL 400, MATVP is used only supported for elements with property extensions. This implies that for such elements, PBAR / PBART, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATVP data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
11. When used with PBAR, PBEAM, PBEAML, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL or PLPLANE (with BEH=PSTRS) the explicit formulation must be used, set VALC3=0 on NLMOPTS.
12. To activating creep analysis with ANAND creep model, the following parameters are mandatory:
  - a. In Bulk Data Card NLMOPTS: keyword "CREEP" and "LRGSTRN" must be selected with value:  
"LRGSTRN" with value 1 for updated Lagrange formulation, Hypo-elasticity and additive plasticity with mean normal value.  
"CREEP" with valc3=1, Implicit Creep on; and valc4=0/1/2, for elastic tangent as default (Blank or 0), 1 for secant tangent, and 2 for radial setting, respectively.
  - b. In Bulk Data Card MATEP: keyword "IMPCREEP" is required for ANAND model.
  - c. In Bulk Data Card NLSTEP: CREEP in "GENERAL" is required for ANAND creep model.
  - d. In Bulk Data Card MATVP: material parameters of ANAND creep model are defined in MATVP card with keyword ANAND" in the 3<sup>rd</sup> field and following 9 fields.
13. NLSTRESS in Case Control will output the equivalent creep strain on grid point. To output the component of the creep strain, NLOUT in NLSTRESS Case Control command must be set and request Creep output in Bulk Data Card.

In case control section:

```
NLSTRESS (NLOUT=num) = ALL
```

In bulk data section:

```
NLOUT, num, TOTTEMP, CCRPSTRN
```

#### Note:

1. This Bulk Data entry accommodates Marc's input data under the model definition option CREEP.

**MAUXCMD****Auxiliary Command to Spawn one Nastran Process from Another - SOL 600**

Defines auxiliary command to spawn on Nastran process from another Nastran process in SOL 600.

**Format:**

| 1       | 2       | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|---------|---------|------|------|------|------|------|------|------|----|
| MAUXCMD | Command | Cont |    |

**Example:**

|         |                                  |  |
|---------|----------------------------------|--|
| MAUXCMD | /mscinst/nast2008/bin/mdnast2008 |  |
|---------|----------------------------------|--|

| Descriptor | Meaning  |
|------------|--|
| Command    | Enter the command you normally use to execute Nastran from the command line<br>(Character; no default; limit of 64 characters) |

**Remark:**

1. At present, this entry may only be used for CINTC when PARAM,MBLADCMD,10 is set.

**MBOLT**      Defines a Bolt For Use in SOL 600 in Countries Outside the USA

Defines a bolt for use in SOL 600 in countries outside the USA. Used in SOL 600 only.

**Format:**

| 1     | 2     | 3     | 4  | 5    | 6  | 7  | 8  | 9  | 10 |
|-------|-------|-------|----|------|----|----|----|----|----|
| MBOLT | ID    | GRIDC | V1 | V2   | V3 |    |    |    |    |
|       | GRIDS | G1    | G2 | G3   | G4 | G5 | G6 | G7 |    |
|       |       | G8    | G9 | Etc. |    |    |    |    |    |
|       | ELEMS | E1    | E2 | E3   | E4 | E5 | E6 | E7 |    |
|       |       | E8    | E9 | Etc. |    |    |    |    |    |

**Example:**

|       |       |      |     |     |     |     |   |   |  |
|-------|-------|------|-----|-----|-----|-----|---|---|--|
| MBOLT | 100   | 1025 | 0.0 | 1.0 | 0.0 |     |   |   |  |
|       | GRIDS | 101  | 102 | 103 | 104 | 105 |   |   |  |
|       | ELEMS | 1    | 2   | 3   | 4   | 5   | 6 | 7 |  |
|       |       | 10   |     |     |     |     |   |   |  |

| Descriptor   | Meaning  |
|--------------|--|
| ID           | Element ID of the bolt. (Integer; Required; no Default)  |
| GRIDC        | Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)   |
| V1           | First component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)   |
| V2           | Second component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)  |
| V3           | Third component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)   |
| GRIDS        | Enter the character string GRIDS to define the start of the entry that defines all of the grids at the bolt intersection cross section (do not enter the ID for GRIDC). (Integer; no Default)  |
| G1, G2, etc. | Grid IDs of the grid points at the bolt intersection. (Integer; no Default)  |
| ELEMS        | Enter the character string ELEMS to define the start of the entry that defines all of the elements at the bolt intersection cross section lying on the side of the cross section corresponding to the negative normal direction. (Integer; no Default) |
| E1, E2, etc. | Element IDs of the grid points at the bolt intersection. (Integer; no Default)   |

**Remarks:**

1. Enter as many GRIDS and ELEMS lines as necessary to define all the grid and element IDs in the cross section.
2. All GRIDS must precede all ELEMS.
3. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersecting surfaces are described by this entry.
4. Specify a different MBOLT entry for each individual bolt.
5. This entry can only be used with Marc 2003 or later outside the USA.
6. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation).
7. This entry maps to Marc's CROSS SECTION entry.
8. GRIDC must already exist. It is not generated by the MBOLT entry. It typically would not be used by any other element, MPC, etc.

**MBOLTUS**

Defines a Bolt for Use in SOL 600 in the USA

Defines a bolt for use only in SOL 600 and only in the USA.

**Format:**

| 1       | 2      | 3     | 4   | 5    | 6   | 7   | 8   | 9   | 10 |
|---------|--------|-------|-----|------|-----|-----|-----|-----|----|
| MBOLTUS | ID     | GRIDC |     |      |     |     |     |     |    |
|         | TOP    | GT1   | GT2 | GT3  | GT4 | GT5 | GT6 | GT7 |    |
|         |        | GT8   | GT9 | Etc. |     |     |     |     |    |
|         | BOTTOM | GB1   | BG2 | BG3  | BG4 | GB5 | GB6 | GB7 |    |
|         |        | GB8   | GB9 | Etc. |     |     |     |     |    |

**Example:**

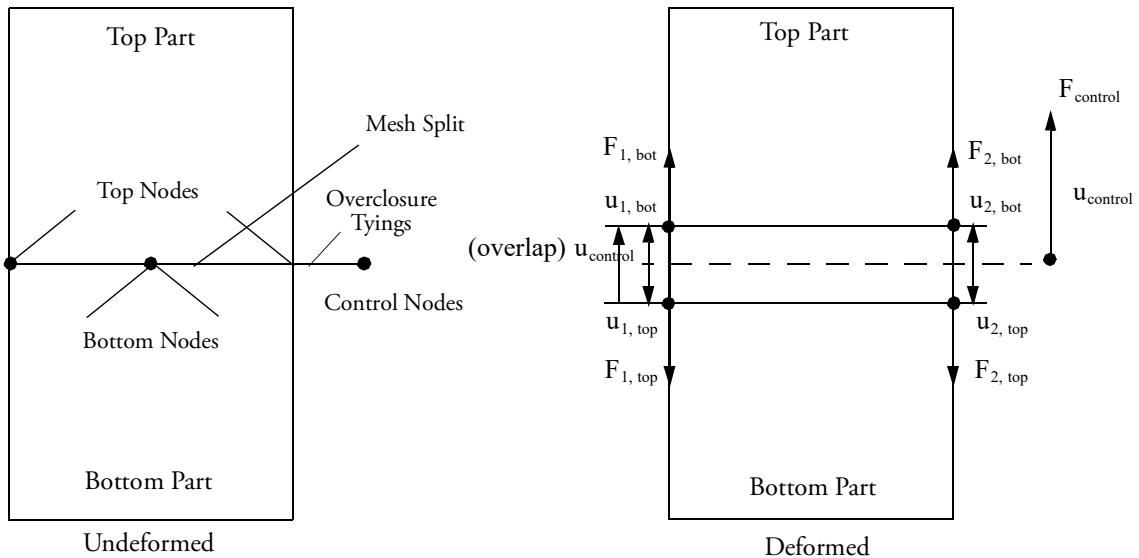
|         |        |      |     |     |     |     |  |  |  |
|---------|--------|------|-----|-----|-----|-----|--|--|--|
| MBOLTUS | 100    | 1025 |     | 1.0 |     |     |  |  |  |
|         | TOP    | 101  | 102 | 103 | 104 | 105 |  |  |  |
|         | BOTTOM | 1    | 2   | 3   | 4   | 5   |  |  |  |

| Descriptor     | Meaning  |
|----------------|--|
| ID             | Element ID of the bolt. (Integer; Required; no Default)  |
| GRIDC          | Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)   |
| TOP            | Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)      |
| GT1, GT2, etc. | Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)   |
| BOTTOM         | Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC) (Integer; no Default) |
| GB1, GB2, etc. | Grid IDs of the grid points at the bottom of the bolt intersection (Integer; no Default)   |

**Remarks:**

1. Enter as many GRIDS as necessary (up to the limit of 999 top grids and 999 bottom grids) to define all the grids at the “top” and “bottom” of the bolt intersection with the structure.
2. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersection surfaces are described by the entry.
3. Specify a different MBOLTUS entry for each individual bolt.
4. This entry can only be used with Marc 2003 or later.

5. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation)
6. This entry maps to Marc's TYING type 69.
7. GRIDC must already exist. It is not generated by the MBOLTUS entry. It typically would not be used by any other element, MPC, etc.
8. The following figure indicates the required modeling and data input.



**MCHSTAT**

## Option to Change State Variables for SOL 600

This option provides various ways of changing state variables throughout the model. It is required if Bulk Data entry, MTCREEP is used. It may also be used to enter temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file in which case MINSTAT is used to define the initial stress-free temperatures and MCHSTAT is used to define the temperatures that cause thermal strains.

**Format:**

| 1       | 2       | 3    | 4    | 5    | 6    | 7    | 8     | 9    | 10 |
|---------|---------|------|------|------|------|------|-------|------|----|
| MCHSTAT | ID      | IDV  | IOPT |      | INCR | NSET | IFORM | IPRT |    |
|         |         | NPST |      |      |      |      |       |      |    |
|         | “FILE”  | Name |      |      |      |      |       |      |    |
|         | “ELEM”  | ELE1 | ELE2 | INT1 | INT2 | LAY1 | LAY2  | VAL  |    |
|         | “STATE” | NS   | IS1  | IS2  | IS4  | IS5  | IS6   |      |    |
|         |         | IS7  | IS8  | IS9  | etc. |      |       |      |    |

**Example:**

|         |      |                        |   |  |   |  |   |  |  |
|---------|------|------------------------|---|--|---|--|---|--|--|
| MCHSTAT | 0    | 1                      | 3 |  | 1 |  | 0 |  |  |
|         |      | 9                      |   |  |   |  |   |  |  |
|         | FILE | change_state_example01 |   |  |   |  |   |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | ID of a matching Case Control MCHSTAT command. If ID=0 this entry is in the Marc Model definition, otherwise it is in Marc's History Definition for the applicable subcase. (Integer; no Default)   |
| IDV        | State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.  |
| IOPT       | Option of how to enter the data. (Integer; Default = 3) (2,2) <ul style="list-style-type: none"> <li>1      Use the “ELEM” continuation line for as many elements as necessary</li> <li>2      Enter the data using user subroutine NEWSV</li> <li>3      Read the data from a t16 or t19 file (see IFORM)</li> </ul> |
| INCR       | Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)  |
| NSET       | Number of sets to be read to define the temperature history if Marc's iteration method is controlled using MTHERM or MTCREEP. (Integer; no Default) (2,6)   |
| IFORM      | Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) <ul style="list-style-type: none"> <li>0      Use binary (t16) file</li> </ul>   |

| Descriptor | Meaning  |
|------------|--|
|            | 1      Use formatted (t19) file  |
| IPRT       | Enter a value of 1 to suppress printing of state variable values defined in user subroutine NEWSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)  |
| NPST       | Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9) See MARCOUT for a list of the post codes.   |
| FILE       | Enter the character string FILE if IOPT=3. (Character, no Default; Required if IOPT=3)   |
| NAME       | Enter the file root name of the previous heat transfer job without any extension. For example if the previous heat transfer job was heat32.dat or heat32.bdf, enter heat32. The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file. (Character, no Default) |
| ELEM       | Enter the character string ELEM to start a list of elements and associated values if IOPT=1 (Character)  |
| ELE1       | First element with value VAL. (Integer; no Default; Required) (3,1)  |
| ELE2       | Last element with value VAL. (Integer; Default = ELE1) (3,2)   |
| INT1       | First integration point with value VAL. (Integer; no Default; Required) (3,3)  |
| INT2       | Last integration point with value VAL. (Integer; Default = INT1) (3,4)   |
| LAY1       | First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)  |
| LAY2       | Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)   |
| VAL        | New state value for these elements. (Real; no Default; Required) (4,1)   |
| STATE      | Enter the character string STATE to start a list of state variables. (Character)   |
| NS         | Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)  |
| ISi        | State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.  |

**Remarks:**

1. All MCHSTAT ID's must be unique.
2. This entry maps to Marc's CHANGE STATE entry.
3. This entry must be entered if the MTCREEP (Marc's AUTO THERM CREEP) entry is used.
4. (i,j) refer to Marc's CHANGE STATE (data block, field).
5. MCHSTAT (and/or MINSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a Case Control LOAD command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a Case Control LOAD command to reference it.

**MCOHE****Interface Cohesive Zone Modeling Element Material Properties**

Specifies material cohesive properties for a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400 only.

**Format:**

| 1     | 2    | 3     | 4     | 5    | 6   | 7   | 8    | 9   | 10 |
|-------|------|-------|-------|------|-----|-----|------|-----|----|
| MCOHE | MID  | MODEL |       | TID  |     |     |      |     |    |
|       | COHE | CRTOD | MAXOD | SNSR | EXP | VED | RRRD | SFC |    |
|       |      |       |       |      |     |     |      |     |    |

**Example:**

|       |       |     |     |     |      |     |  |     |  |
|-------|-------|-----|-----|-----|------|-----|--|-----|--|
| MCOHE | 701   | 2   |     | 357 |      |     |  |     |  |
|       | 136.5 | 0.0 | 0.0 | 1.0 | 0.02 | 0.0 |  | 1.0 |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Identification number of a MCOHE entry. (Integer > 0) (See Remark 1.)   |
| MODEL      | (Integer > 0; Default = 1) (See Remark 3.)  |
|            | 1      Bilinear model   |
|            | 2      Exponential model  |
|            | 3      Linear-exponential model   |
|            | -1     User Defined Subroutine (See Remark 2.)  |
| TID        | Table identifier for a combination of TABLES1/TABLEST for cohesive energy vs temperature. (Integer $\geq 0$ ; Default = 0)  |
| COHE       | Cohesive energy. (Real $\geq 0.0$ )   |
| CRTOD      | Critical opening distance. (Real $\geq 0.0$ )   |
| MAXOD      | Maximum opening displacement (bilinear model only). (Real $\geq 0.0$ )  |
| SNSR       | Shear Normal Stress Ratio. (Real $> 0.0$ ; Default = 1.0)   |
| EXP        | Exponential decay factor (linear-exponential model only). (Real $> 0.0$ ; Default = 1.0)  |
| VED        | Factor for viscous energy dissipation. (Real $\geq 0.0$ ; Default = 0.0)  |
| RRRD       | Reference rate of relative displacement. Used only if VED $\neq 0.0$ . A value of 0.0 implies that the reference rate will be automatically calculated. (Real $\geq 0.0$ ; Default = 0.0) |
| SFC        | Stiffening factor in compression. (Real $\geq 0.0$ ; Default = 1.0)   |
| SNER       | Shear Normal Energy Ratio. (Real $> 0.0$ ; Default = 1.0)   |

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. User subroutine UCOHES is used. User must also use the MATUDS bulk data entry.
3. For details on model see Nonlinear User's Guide (SOL 400).

**MDBCNCT**

Defines the Touching and Touched Contact Bodies in Different Modules

Defines the touching and touched contact and used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, and 112 for general contact, glued, step glue or permanent glue between contact bodies residing in different Modules.

**Format:**

| 1       | 2         | 3      | 4      | 5      | 6       | 7     | 8        | 9 | 10 |
|---------|-----------|--------|--------|--------|---------|-------|----------|---|----|
| MDBCNCT | ID        | BCGPID | BCPPID | MODS   | IDSLAVE | MODM  | IDMASTER |   |    |
|         | “SLAVES”  | MODS1  | IDSL1  | MODS2  | IDSL2   | MODS3 | IDSL3    |   |    |
|         |           | MODS4  | IDSL4  | -etc.- |         |       |          |   |    |
|         | “MASTERS” | MODM1  | IDMA1  | MODM2  | IDMA2   | MODM3 | IDMA3    |   |    |
|         |           | MODM4  | IDMA4  | -etc.- |         |       |          |   |    |

**Example:**

|         |    |     |  |     |   |     |      |  |  |
|---------|----|-----|--|-----|---|-----|------|--|--|
| MDBCNCT | 57 | 306 |  | 101 | 2 | 201 | 1002 |  |  |
|---------|----|-----|--|-----|---|-----|------|--|--|

|         |         |     |     |     |     |     |     |  |  |
|---------|---------|-----|-----|-----|-----|-----|-----|--|--|
| MDBCNCT | 9       |     | 108 |     |     |     |     |  |  |
|         | SLAVES  | 101 | 30  | 201 | 26  |     |     |  |  |
|         | MASTERS | 101 | 294 | 201 | 135 | 301 | 528 |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number referenced by a MDBCTB1 entry (Integer $\geq 0$ ). See Remark 5.                        |
| BCGPID     | Parameter identification number of a BCONPRG entry (Integer $\geq 0$ or blank). See Remark 6.                        |
| BCPPID     | Parameter identification number of a BCONPRP entry (Integer $\geq 0$ or blank). See Remark 6.                        |
| MODS       | Identification number of a Module that defines IDSLAVE (Integer $\geq 0$ or blank)                                   |
| IDSLAVE    | Identification number of BCBODY1 entry defining the touching body (Integer $\geq 0$ or blank). See Remarks 7. and 8. |
| MODM       | Identification number of a Module that defines IDMASTER (Integer $\geq 0$ or blank)                                  |
| IDMASTER   | Identification number of BCBODY1 entry defining the touched body (Integer $\geq 0$ or blank). See Remarks 9. and 10. |
| “SLAVES”   | Indicates the start of the list of the touching bodies. See Remark 8.  |
| MODSi      | Identification number of a Module that defines IDSLi (Integer $\geq 0$ or blank)                                     |
| IDSLi      | Identification number of BCBODY1 entry defining touching bodies (Integer $\geq 0$ or blank,). See Remarks 7. and 8.  |

| Descriptor | Meaning   |
|------------|---|
| "MASTERS"  | Indicates the start of the list of bodies touched by touching bodies. See Remark 10.        |
| MODMi      | Identification number of a Module that defines IDMAi (Integer $\geq 0$ or blank)            |
| IDMAi      | Identification number of BCBODY1 entry defining touched bodies (Integer $\geq 0$ or blank). |

#### Remarks:

1. Both BCTABL1 and MDBCTB1 may only be specified in the main Bulk Data section.
2. BCTABL1 BCID must be unique with respect to all other BCTABL1 and MDBCTB1.
3. BCONTACT Case Control command may only select BCTABL1 or MDBCTB1 but not both. If BCTABL1 is selected, then all bodies matching IDi in any and all Modules will be activated.
4. Only MDBCTB1 can select MDBCNCT.
5. MDBCNCT can be selected by the Case Control command BCONTACT=ID to define surface contact if MDBCTB1 entry does not exist. See Remarks 2 and 3 of MDBCTB1 entry.
6. If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
7. A short input to define a single touching body exists if the user provides IDSLAVE. On the other hand, if the user leaves IDSLAVE blank, then "SLAVES" descriptor is required and IDSL1 must be specified..
8. "SLAVES" and IDSLi fields will be ignored if IDSLAVE exists. If IDSLAVE field is blank, then "SLAVES" and IDSLi must be specified. In this case, each IDSLi will be processed separately.
9. A short input to define a single touched body exists if the user provides IDMASTER. On the other hand, if the user leaves IDMASTER blank, then "MASTERS" descriptor is required and IDMA1 must be specified.
10. "MASTERS" and IDMAi fields will be ignored if IDMASTER exists. If IDMASTER field is blank, then "MASTERS" and IDMAi must be specified.
11. The concept of Slave and Master relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
12. If all the BCONPRG that are referenced by a MDBCNCT (which is referenced by MDBCTB1) have a value of IGLUE > 0, and this MDBCNCT is referenced in the 1st Loadcase (SOL 100\*) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

**MDBCTB1**

Defines a Contact Table for Bodies in Different Modules

MDBCTB1 specifies a list of contact pairs (for bodies residing in different Modules) through the MDBCNCT option for SOL 101. This contact table is activated in the BCONTACT Case Control command.

**Format:**

| 1       | 2    | 3    | 4   | 5      | 6   | 7    | 8   | 9    | 10 |
|---------|------|------|-----|--------|-----|------|-----|------|----|
| MDBCTB1 | BCID | MOD1 | ID1 | MOD2   | ID2 | MOD3 | ID3 | MOD4 |    |
|         | ID4  | MOD5 | ID5 | -etc.- |     |      |     |      |    |

**Examples:**

|         |     |      |     |      |     |     |     |     |  |
|---------|-----|------|-----|------|-----|-----|-----|-----|--|
| MDBCTB1 | 2   | 101  | 198 | 201  | 62  | 301 | 75  | 401 |  |
|         | 501 | 8    | 601 | 159  | 701 | 31  | 801 | 44  |  |
| MDBCTB1 | 0   | 1001 | 23  | 1002 | 56  |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| BCID       | Unique identification number referenced by a BCONTACT Case Control command.<br><br>See Remark 3.(Integer $\geq 0$ ; Required) |
| MODi       | Identification number of module that defines BCNECT IDi (Integer $\geq 0$ or blank,)  |
| IDi        | Identification number of BCNECT entry (Integer $> 0$ ). See Remark 4. and 5.  |

**Remarks:**

1. MDBCTB1 defines surface contact. If MDBCTB1 does not exist, the Case Control command BCONTACT=BCID may refer to the MDBCNCT Bulk Data entry directly.
2. If neither MDBCTB1 nor MDBCNCT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If MDBCTB1 or MDBCNCT is given, the default for everybody is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. MDBCTB1 or MDBCNCT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. For SOL 101, the MDBCTB1 or MDBCNCT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY1's. To place an entry in

any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or MDBCNCT=BCID. When BCONTACT=ALLBODY, there is no BCID of MDBCTB1 or MDBCNCT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.

4. The “THRU” option is not supported.
5. Intervening blank fields are not allowed.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**MDBNDRY****Module to Module Boundary Point Definitions**

Defines a list of grid points in a module for the automatic boundary search between a specified Module or between all other Modules. In other words, the automatic boundary search will only search for connections in this list.

**Format:**

| 1       | 2     | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|-------|-------|-------|-------|-------|-------|----|
| MDBNDRY | MIDA  | MIDB  | GIDA1 | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | etc.  |       |       |       |       |       |    |

**Examples 1:**

|         |     |   |    |    |    |    |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|
| MDBNDRY | 400 | 4 | 10 | 20 | 30 | 40 |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|

**Examples 2:**

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| MDBNDRY | 400 | ALL | 10 | 20 | 30 | THRU | 35 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| MIDA       | Module identification number. (Integer $\geq 0$ , Default=0)  |
| MIDB       | Reference Module identification number. (Integer $\geq 0$ or Character “ALL”; Default=ALL)                                    |
| GIDAi      | Identification number of a boundary grid point in the Module MIDA. (Integer $> 0$ or “THRU”; for “THRU” option, GID1 < GID2.) |

**Remarks:**

1. MDBNDRY may only be specified in the Main Bulk Data Section and cannot be specified in other BEGIN MODULE sections.
2. This entry is valid only if Modules exist.

**MDBOLT**

Defines the Multi-Point Constraints for a Bolt Between Two Modules

Defines a rigid bolt between two Modules by a set of MPC constraints.

Format:

| 1      | 2      | 3     | 4    | 5    | 6    | 7   | 8   | 9 | 10 |
|--------|--------|-------|------|------|------|-----|-----|---|----|
| MDBOLT | ID     | GRIDC | MODC | MODT | MODB |     |     |   |    |
|        | TOP    | GT1   | GT2  | GT3  | GT4  | GT5 | GT6 |   |    |
|        |        | GT8   | GT9  | etc. |      |     |     |   |    |
|        | BOTTOM | GB1   | GB2  | GB3  | BG4  | GB5 | GB6 |   |    |
|        |        | GB8   | GB9  | etc. |      |     |     |   |    |

Example:

|        |        |      |     |     |     |     |  |  |  |
|--------|--------|------|-----|-----|-----|-----|--|--|--|
| MDBOLT | 100    | 1025 | 11  | 11  | 30  |     |  |  |  |
|        | TOP    | 101  | 102 | 103 | 104 | 105 |  |  |  |
|        | BOTTOM | 1    | 2   | 3   | 4   | 5   |  |  |  |

| Descriptor        | Meaning   |
|-------------------|---|
| ID                | Element ID of the bolt. (Integer; Required; no Default)   |
| GRIDC             | Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)  |
| TOP               | Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)                                       |
| MODC              | Module containing the GRID entry for GRIDC. (Integer; Default=0, If MODC>0 then must be same as MODT or MODB)   |
| MODT              | Module containing the GRID entries for GTi. (Integer; Default=0)  |
| MODB              | Module containing the GRID entries for GBi. (Integer; Default=0)  |
| GT1, GT2,<br>etc. | Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)  |
| BOTTOM            | Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default) |
| GB1, GB2,<br>etc. | Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)   |

Remarks:

1. The MDBOLT entry must be defined in the main Bulk Data section only (Module 0).

2. See all remarks and figures under the BOLT entry description. All remarks and figures under BOLT apply to MDBOLT.

**MDBULK****Module Type Definitions**

Defines module search options and typing such as repeated or mirrored.

**Format:**

| 1      | 2     | 3    | 4      | 5      | 6   | 7 | 8 | 9 | 10 |
|--------|-------|------|--------|--------|-----|---|---|---|----|
| MDBULK | MODID | TYPE | RMODID | METHOD | TOL |   |   |   |    |

**Example:**

|        |    |  |  |      |        |  |  |  |  |
|--------|----|--|--|------|--------|--|--|--|--|
| MDBULK | 14 |  |  | AUTO | 1.0e-3 |  |  |  |  |
|--------|----|--|--|------|--------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MODID      | Module identification number. (Integer > 0 or Character = "ALL")   |
| TYPE       | Module type (Character; default = "PRIMARY") <ul style="list-style-type: none"> <li>PRIMARY Module with at least one GRID entry in its BEGIN MODULE section.</li> <li>REPEAT Copied Module via reposition or mirroring of a primary Module. See Remarks 9 and 10.</li> <li>MIRROR Copied Module via mirroring of a primary Module. See Remarks 9 and 10.</li> <li>MOVE Copied or primary Module repositioning via MDMOVE entry. See Remarks 8 and 10.</li> </ul> |
| RMODID     | Identification number of the reference Module, used only if making a copy; i.e., RMODID>0 and RMODID?MODID and TYPE= "REPEAT", "MIRROR" or "MOVE". (Integer>0; Default=0). See Remarks 9 and 10.   |
| METHOD     | Method to be used when searching for boundary grid points. (Character = "AUTO" or "MANUAL"; Default= "MANUAL")   |
| TOL        | Location tolerance to be used when search for boundary grid points.<br>(Real; Default = 1.0e-5)  |

**Remarks:**

1. METHOD="MANUAL" requires MDCONCT entries. MDBNDRY and MDEXCLD, that reference MODID, will produce fatal messages.
2. MDCONCT, MDBNDRY and MDEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
3. For combined automatic and manual boundary search, the METHOD="AUTO" should be specified and connections should be specified on a MDCONCT entry.
4. TOL is the default value that can be modified between two Modules by providing the required tolerance on the MDCONCT entry.

5. MDBULK may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections
6. This entry is only valid if Modules exist.
7. With MDBULK,ALL the TYPE and RMODID fields must be blank and MDBULK,ALL means that the MDBULK options METHOD and TOL will be applied to all Modules which are not already defined on an MDBULK entry.
8. TYPE="MOVE" refers to the enhanced interface. If TYPE="MOVE" then an MDMOVE Bulk Data entry must be defined. The MDMOVE entry references MDTRAN, MDROTi, and MDMIRi entries.
  - a. If TYPE="MOVE" is specified for a given Module, then MDLOC and MDMPLN cannot be specified and vice-versa.
  - b. If MDMIRi is referenced by MDMOVE then it must be the first MVID and cannot be specified as the second or subsequent MVIDs.
  - c. If MODID defines a secondary Module (RMODID>0) then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
  - d. A primary Module (RMODID=0) may be re-positioned as indicated in Remark 10.
9. TYPE="REPEAT" and "MIRROR" refer to the classic interface. These types have many common features, but they also have some important differences. These are described below:
  - a. The following comments apply to both TYPE="REPEAT" and "MIRROR":
    - A Module (MODID) whose TYPE is "REPEAT" or "MIRROR" and RMODID is greater than 0 is referred to as a secondary Module. The RMODID specified in this case is regarded as the primary Module.
    - The primary Module may be re-positioned as indicated in Remark 10.
    - If MODID defines a secondary Module then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
    - If both an MDLOC and MDMPLN entry are specified for a secondary Module, then a mirror image copy of the primary Module will be created by first using the plane defined by the MDMPLN entry and then repositioned at the location implied by the MDLOC entry.
  - b. The following comments apply only to TYPE= "REPEAT":
    - The secondary Module in this case may reference an MDLOC entry, an MDMPLN entry or both or none.
    - If it references an MDLOC entry, then an identical copy of its primary Module will be positioned at the location implied by the MDLOC entry.
    - If it references an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.

- If it references neither an MDLOC entry nor an MDMPLN entry, then the secondary Module will merely be a duplicate of the primary Module positioned at the same location as the primary Module. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message
  - c. The following comments apply only to TYPE= "MIRROR":
    - The secondary Module in this case must reference an MDMPLN. (Otherwise, the program will terminate the execution with an appropriate user fatal message.)
    - If the secondary Module references only an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.
10. A Module whose TYPE is "PRIMARY" (or TYPE="MOVE" and RMODID=0) may itself be repositioned by the use of MDLOC or MDMPLN entry or both (or MDTRAN, MDROTi or MDIRi or any combination).
- a. If it references an MDLOC (or MDTRAN, MDROTi) entry, then the primary Module will be positioned at the location implied by the MDLOC (or MDTRAN, MDROTi) entry.
  - b. If it references an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will be repositioned using the plane defined by the MDMPLN (or MDMIRi) entry.
  - c. If it references both an MDLOC (or MDTRAN, MDROTi) entry and an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will first be created using the plane defined by the MDMPLN (or MDMIRi) entry and then repositioned at the location specified by the MDLOC (or MDTRAN, MDROTi) entry.

**MDCONCT****Module Boundary Point Connections**

Explicitly defines grid and scalar connection procedures for a module to module operation.

**Format:**

| 1       | 2    | 3    | 4    | 5    | 6    | 7    | 8     | 9    | 10 |
|---------|------|------|------|------|------|------|-------|------|----|
| MDCONCT | BID  | TYPE | TOL  | X    | Y    | Z    | MODID | CID  |    |
|         | MID1 | GID1 | MID2 | GID2 | MID3 | GID3 | MID4  | GID4 |    |
|         | MID5 | GID5 | etc. |      |      |      |       |      |    |

**Alternate Format:**

| 1       | 2    | 3    | 4    | 5    | 6    | 7    | 8     | 9    | 10 |
|---------|------|------|------|------|------|------|-------|------|----|
| MDCONCT | BID  | TYPE | TOL  | GRID |      |      | MODID |      |    |
|         | MID1 | GID1 | MID2 | GID2 | MID3 | GID3 | MID4  | GID4 |    |
|         | MID5 | GID5 | etc. |      |      |      |       |      |    |

**Example 1:**

|         |     |     |     |     |       |       |  |  |  |
|---------|-----|-----|-----|-----|-------|-------|--|--|--|
| MDCONCT | 123 |     | 0.4 | 0.5 | 74.12 | 100.3 |  |  |  |
|         | 10  | 423 | 20  | 123 |       |       |  |  |  |

**Example 2:**

|         |     |       |     |     |  |  |  |  |  |
|---------|-----|-------|-----|-----|--|--|--|--|--|
| MDCONCT | 123 | RRBE2 | 0.4 | 777 |  |  |  |  |  |
|         | 10  |       | 20  |     |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| BID        | Unique Boundary identification number. (Integer > 0)  |
| TYPE       | Connection Option. (Character: “MERGE”, “RIGID”, “RRBE2”, “MRBE2” or Blank, Default=“MRBE2”).             |
| TOL        | Location tolerance to be used when searching or checking for boundary grid points. (Real; Default=1.0e-5) |
| X,Y,Z      | Connection (Search) location. (Real, Default=0.0)   |
| GRID       | Grid Identification for Connection (Search). (Integer > 0) See Remark 11.                                 |
| MODID      | Module Identification number for the Location Reference. (Integer ≥ 0, Default=0)                         |
| CID        | Coordinate System in MODID to be applied to Location. (Integer ≥ 0, Default=0)                            |

| Descriptor | Meaning  |
|------------|--|
| MIDI       | Module Identification number for grid/scalar reference. (Integer $\geq 0$ )  |
| GIDi       | Identification number of a grid or scalar point in the reference module MIDI, which will be verified to be in TOL of the Location. (Integer $\geq 0$ ) |

**Remarks:**

1. MDCONCT may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.
2. TOL can be used to override the default value specified on the MDBULK entries.
3. The continuation entry is optional when used in automatic searching.
4. The GIDi selections must be all of the same class, grids or scalar points.
5. All six degrees-of-freedom of grids will be defined as boundary degrees-of-freedom.
6. This entry is only valid if Modules exist.
7. When GIDi is blank or zero, then all grids of the module will be tested against the location and tolerance.
8. MIDI values can be specified more than once.
9. For TYPE=“MERGE”, the boundary grids will be moved to the location specified by MODID and connected by RBAR when only two boundary nodes are specified or RBE2 when more than two boundary nodes are specified.
10. For TYPE=“MRBE2”, the boundary grids will be moved to the location specified by MODID and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
11. For TYPE=“RIGID”, the boundary grids will not be moved and will be connected via RBE2 elements.
12. For TYPE=“RRBE2”, the boundary grids will not be moved and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
13. When GRID is used, then this identification/location will be used for the independent point of the RBE2 element.
14. When scalar points are specified for GIDi, then GRID, X, Y, Z, MODID and CID should not be specified.
15. If any GIDi are not connected due to exceeding TOL then User Fatal Message 6783 will be issued.

**MDDMIG****Direct Matrix Input at Points Defined in Two or More Modules**

Defines direct input matrices related to grid, extra, and/or scalar points defined in two or more Modules. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

| 1      | 2    | 3   | 4   | 5   | 6    | 7     | 8 | 9    | 10 |
|--------|------|-----|-----|-----|------|-------|---|------|----|
| MDDMIG | NAME | "0" | IFO | TIN | TOUT | POLAR |   | NCOL |    |

**Column Entry Format:**

|        |      |      |    |    |    |    |        |  |  |
|--------|------|------|----|----|----|----|--------|--|--|
| MDDMIG | NAME | MODJ | GJ | CJ |    |    |        |  |  |
|        |      | MOD1 | G1 | C1 | A1 | B1 |        |  |  |
|        |      | MOD2 | G2 | C2 | A2 | B2 | -etc.- |  |  |

**Example:**

|        |      |    |    |   |        |     |  |  |  |
|--------|------|----|----|---|--------|-----|--|--|--|
| MDDMIG | STIF | 0  | 1  | 3 | 4      |     |  |  |  |
| MDDMIG | STIF | 11 | 27 | 1 |        |     |  |  |  |
|        |      | 20 | 2  | 3 | 3.+5   | 3+3 |  |  |  |
|        |      | 20 | 2  | 4 | 2.5+10 | 0.  |  |  |  |
|        |      | 45 | 50 |   | 1.0    | 0.  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| NAME       | Name of the matrix. See Remark 1 under DMIG. (One to eight alphanumeric characters, the first of which is alphabetic.)             |
| IFO        | Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) |
|            | 1            Square  |
|            | 2 or 9      Rectangular  |
|            | 6            Symmetric   |
| TIN        | Type of matrix being input: (Integer)  |
|            | 1            Real, single precision (One field is used per element.)   |
|            | 2            Real, double precision (One field is used per element.)   |
|            | 3            Complex, single precision (Two fields are used per element.)  |
|            | 4            Complex, double precision (Two fields are used per element.)  |
| TOUT       | Type of matrix that will be created: (Integer)   |

| Descriptor | Meaning  |
|------------|--|
| 0          | Set by precision system cell (Default)   |
| 1          | Real, single precision   |
| 2          | Real, double precision   |
| 3          | Complex, single precision  |
| 4          | Complex, double precision  |
| POLAR      | Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)              |
| NCOL       | Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5 and 6 under DMIG. (Integer > 0)                            |
| MODJ       | Module IDs that contain the GRID entry for GJ. (Integer $\geq 0$ ).  |
| GJ         | Grid, scalar or extra point identification number in Module MODJ for column index. (Integer > 0)   |
| CJ         | Component number for grid point GJ. (0 < Integer < 6; blank or zero if GJ is a scalar or extra point.)                                     |
| MODi       | Module IDs that contain the GRID entry for Gi. (Integer $\geq 0$ ).  |
| Gi         | Grid, scalar, or extra point identification number in Module MODi for row index. (Integer > 0)   |
| Ci         | Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)                                    |
| Ai, Bi     | Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real) |

**Remarks:**

1. The MDDMIG entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the DMIG entry description. All remarks and figures under DMIG apply to MDDMIG.
3. When IFO=2 or 9, MODJ indicates the column number and not a module ID. GJ and CJ are ignored.

**MDELAM**

## Delamination materials in SOL 600

Defines materials for which delamination may occur in SOL 600 only.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6    | 7    | 8    | 9    | 10 |
|--------|--------|--------|--------|--------|------|------|------|------|----|
| MDELAM |        | MID1   | MID2   | ITYPE  | STRN | EXPN | STRT | EXPT |    |
|        | ITSTRN | ITEXPN | ITSTRT | ITEXPT |      |      |      |      |    |

**Example:**

|        |  |   |   |   |        |    |        |    |  |
|--------|--|---|---|---|--------|----|--------|----|--|
| MDELAM |  | 4 | 5 | 2 | 35000. | 1. | 20000. | 1. |  |
|--------|--|---|---|---|--------|----|--------|----|--|

| Descriptor | Meaning   |
|------------|---|
| MID1       | First material ID. (Integer; no Default; required field)  |
| MID2       | Second material ID. If MID2=MID1 delamination occurs within the material. If MID2 is not the same as MID1, delamination may occur at the interfaces between the materials. (Integer; Default = MID1)                              |
| ITYPE      | Delamination type (Integer; Default = 1) <ul style="list-style-type: none"> <li>1      Determine if delamination occurs but do not actually allow delamination</li> <li>2      Split the mesh when delamination occurs</li> </ul> |
| STRN       | Allowable normal stress (Real; no Default; required value)  |
| EXPN       | Exponent for normal stress (Real; Default = 1.0)  |
| STRT       | Allowable tangential stress (Real; no Default; required value)  |
| EXPT       | Exponent for tangential stress (Real; Default = 1.0)  |
| ITSTRN     | ID of a TABL3D entry providing the variation of STRN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)   |
| ITEXPN     | ID of a TABL3D entry providing the variation of EXPN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)   |
| ITSTRT     | ID of a TABL3D entry providing the variation of STRT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)   |
| ITEXPT     | ID of a TABL3D entry providing the variation of EXPT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)   |

**Remarks:**

1. The delamination criteria is based upon  $(S_n/STRN)^{**EXPB} + (S_t/STRT)^{**EXPT} > 1.0$   
Where  $S_n$  and  $S_t$  are the normal and tangential stresses
2. If no tables are required, the continuation entry may be omitted.

**MDEXCLD****Module to Module Excluded Boundary Point Definitions**

Defines a list of grid points in a Module that will be excluded during the attachment to another Module.

**Format:**

| 1       | 2     | 3     | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|-------|-------|-------|-------|-------|-------|----|
| MDEXCLD | MIDA  | MIDB  | GIDA1 | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | etc.  |       |       |       |       |       |    |

**Example 1:**

|         |     |   |    |    |    |    |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|
| MDEXCLD | 400 | 4 | 10 | 20 | 30 | 40 |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|

**Example 2:**

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| MDEXCLD | 400 | ALL | 10 | 20 | 30 | THRU | 35 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| MIDA       | Module identification number. (Integer $\geq 0$ , Default=0)  |
| MIDB       | Reference Module identification number. (Integer $\geq 0$ or Character "ALL"; Default=ALL)  |
| GIDAi      | Identification number of a boundary grid point in the module MIDA to be excluded from connection to module MIDB. (Integer $> 0$ or "THRU"; for "THRU" option, GID1 < GID2.) |

**Remarks:**

1. This entry is only valid if Modules exist.
2. MDEXCLD may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.

## MDFAST

### A Shell Patch Fastener Connection Between Two Modules

Defines a fastener with material orientation connecting two surface patches defined in two Modules.

**Format:**

| 1      | 2   | 3   | 4    | 5   | 6   | 7    | 8    | 9  | 10 |
|--------|-----|-----|------|-----|-----|------|------|----|----|
| MDFAST | EID | PID | TYPE | IDA | IDB | GS   | GA   | GB |    |
|        | XS  | YS  | ZS   |     |     | MODA | MODB |    |    |

**Example using PROP:**

|        |   |    |      |    |    |     |    |    |  |
|--------|---|----|------|----|----|-----|----|----|--|
| MDFAST | 3 | 20 | PROP | 21 | 24 | 206 |    |    |  |
|        |   |    |      |    |    |     | 10 | 20 |  |

**Example using ELEM:**

|        |   |    |      |    |    |     |    |    |  |
|--------|---|----|------|----|----|-----|----|----|--|
| MDFAST | 7 | 70 | ELEM | 27 | 74 | 707 |    |    |  |
|        |   |    |      |    |    |     | 11 | 12 |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of a PFAST entry. (Integer > 0; Default = EID)   |
| TYPE       | Specifies the surface patch definition: (Character)<br><br>If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28. |
|            | If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28.   |
| IDA, IDB   | Property id (for PROP option) or Element id (for ELEM option) defining patches A and B in Modules MODA and MODB, respectively. IDA≠IDB (Integer > 0)  |
| GS         | Grid point defining the location of the fastener. Its GRID entry must be defined in the main Bulk Data section (Module 0). See also Remark 2. under CFAST. (Integer > 0 or blank)   |
| GA, GB     | Grid ids of piercing points on patches A and B. Their GRID entries must be defined in the main Bulk Data section (Module 0). See also Remark 2 under CFAST. (Integer > 0 or blank)  |

| Descriptor    | Meaning  |
|---------------|--|
| XS,YS,ZS      | Location of the fastener in basic. Required if neither GS nor GA is defined. See also Remark 2. under CFAST. (Real or blank) |
| MODA,M<br>ODB | Module IDs that contain the surface patches defined by property or element entries IDA and IDB (Integer $\geq 0$ ).          |

**Remarks:**

1. The MDFAST entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CFAST entry description. All remarks and figures under CFAST apply to MDFAST.

**MDLABEL****Module Output Label**

Defines a label or name to be printed in the Module's results output page headings.

Format:

| 1       | 2   | 3 | 4 | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|-----|---|---|-------|---|---|---|---|----|
| MDLABEL | MID |   |   | LABEL |   |   |   |   |    |

Example:

|         |     |                                 |  |
|---------|-----|---------------------------------|--|
| MDLABEL | 400 | LEFT REAR FENDER, MODEL XYZ2000 |  |
|---------|-----|---------------------------------|--|

| Descriptor | Meaning  |
|------------|--|
| MID        | Module identification number. (Integer $\geq 0$ , Default=0)                   |
| LABEL      | Label associated with Module MID for results output page headings. (Character) |

Remarks:

1. MDLABEL can only be specified in the main Bulk Data Section and is ignored in-between the BEGIN MODULE=n and ENDMODULE commands.
2. Only one MDLABEL per Module may be specified.
3. The label will appear in all Module's results output page headings. However, in some headings the label may be truncated.
4. This entry is valid only if Modules exist.

**MDLOC****Module Reposition by Translation and/or Rotation**

Defines reposition of a Module by listing three non-collinear points in the Module and three corresponding points in Module 0.

**Format:**

| 1     | 2     | 3   | 4   | 5   | 6   | 7   | 8   | 9 | 10 |
|-------|-------|-----|-----|-----|-----|-----|-----|---|----|
| MDLOC | MODID | PA1 | PA2 | PA3 | PB1 | PB2 | PB3 |   |    |

**Example:**

|       |     |    |     |     |      |     |    |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|
| MDLOC | 110 | 10 | 100 | 111 | 1010 | 112 | 30 |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| MODID      | Module identification number of primary, repeated, or mirrored module (Integer>0).  |
| PAi        | GRID or POINT identification numbers of three non-collinear points in (Integer>0): <ul style="list-style-type: none"> <li>■ "RMODID's Bulk Data section. RMODID is referenced on MDBULK if this MODID is a secondary (copied) Module</li> <li>■ "Module MODID's Bulk Data section if this is a primary Module.</li> </ul> |
| PBi        | GRID or POINT identification numbers of three non-collinear points in Module 0 (main Bulk Data section). Must have the same relative locations as PAi. (Integer>0).   |

**Remarks:**

1. MDLOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. The Module will be rotated and translated for alignment of the PAi and PBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in the BEGIN Module=MODID Bulk Data section.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section or Module 0. If they belong to a Module that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three GRIDs or POINTs are required even if the Module connects to only one or two boundary grids.
8. Coordinate systems, global displacement directions, and element coordinate systems for the Module will rotated and translated.

9. The global coordinate directions of the grid points in the Module will be transformed internally to the global coordinate directions of the grid points in the Module 0. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the Module to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:
  - The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
  - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
  - The z-axis is defined by the cross product of the x-axis into the y-axis.
  - The rotation and translation transformation aligns the local system defined by the Module grids with the local system defined by the main Bulk Data Section grids
11. This entry will only work if Modules (BEGIN MODULE) exist.

**MDLPRM****Model Parameters**

Specifies parameters which affect the solution of the structural model.

**Format:**

| 1      | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9 | 10 |
|--------|--------|------|--------|------|--------|------|--------|---|----|
| MDLPRM | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 | -etc.- |   |    |

**Example:**

|        |        |   |         |   |  |  |  |  |  |
|--------|--------|---|---------|---|--|--|--|--|--|
| MDLPRM | QR6ROT | 2 | QRSHEAR | 1 |  |  |  |  |  |
|--------|--------|---|---------|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PARAMi     | Name of the parameter. Allowable names are given in <a href="#">Table 24</a> . (Character) |
| VALi       | Value of the parameter. (Real or Integer; see <a href="#">Table 24</a> )                   |

**Remark:**

1. Multiple entries of MDLPRM are allowed in the Bulk Data Section. However, multiple entries of a particular parameter PARAMi are illegal.
2. This entry is not supported in SOL 600.
3. If Modules are present then this entry may only be specified in the main Bulk Data section.

**Table 24**    PARAMi Names and Descriptions

| Name   | Description, Type, and Default Values  |  |
|--------|--|--|
| BRTOBM | Flag to determine whether to convert a CBAR element to a CBEAM element for the nonlinear analysis.   |  |
|        | 0  | Do not convert CBAR to CBEAM. (Default)  |
|        | 1  | Convert CBAR to CBEAM.   |
|        | -1   | Same as 1, but print the converted Bulk Data entries on f06 file.  |
| BUSHRT | Flag to control large rotation effects for CBUSH or CFAST elements in nonlinear analysis (SOL400). For CBUSH, this flag is overridden by the LRGR field on the element PBUSHT entry. |  |
|        | 0  | Large rotation effects are included. Element coordinate system is rotated with the rotation of grid A. (Default)   |
|        | 1  | No large rotation effects. Small rotation is used.   |
|        | 2  | Large rotation effects are included. A mid-increment method is used to rotate the element system. This is recommended for large rotation analysis. For the CBUSH element, CID must equal zero. |

Table 24 PARAMi Names and Descriptions

| Name     | Description, Type, and Default Values  |  |
|----------|--|--|
| COMPNI   | Control if stresses and strains of advanced composite elements use MSC Nastran conventional layer format   |  |
|          | NO(default)  | Don't use  |
|          | YES  | Use conventional layer format                                    |
| DBCTOLE  | Tolerance for DBC date conversion, real(default=1.e-4) >= 0.0  |  |
| DCFLTEXP | Determines the exponent of the tolerance value used to filter out small entries from the coefficient matrix before a matrix decomposition. In each column, matrix entries which are DCFLTEXP orders of magnitude smaller than the diagonal entry of the column are filtered out. If DCFLTEXP=0, then the coefficient matrix is not altered. (Integer; Default = 0) |  |
| DELELAS  | Randomly delete $c^*100\%$ of CELASi elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: $c = 0.02$ )  |  |
| DELFAST  | Randomly delete $c^*100\%$ of CFAST elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: $c = 0.02$ )   |  |
| DELMASS  | Randomly delete $c^*100\%$ of CMASSi, CONM1, and CONM2 elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: $c = 0.02$ )  |  |
| DELSEAM  | Randomly delete $c^*100\%$ of CSEAM elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: $c = 0.02$ )   |  |
| DELWELD  | Randomly delete $c^*100\%$ of CWELD elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: $c = 0.02$ )   |  |
| GEV1417  | See Remark 9 of PBUSH  |  |
|          | 0  | Use GE default rules on PBUSH. (Default)                         |
|          | 1  | Use Nastran Version 2014-2017 incompatible rule for GE defaults. |
| GNLSTN   | Strain formulation flag for QUADR/TRIAR elements used in geometric nonlinear analysis. (Parameter LGDISP=1)  |  |
|          | 0  | Small strain. (Default)  |
|          | 1  | Green strain.  |
| HDF5     | Parameter to create NH5RDB database  |  |
|          | -1   | Do not create NH5RDB(default)                                    |
|          | 0  | Create NH5RDB without compression                                |
|          | 1  | Create NH5RDB with compression                                   |
|          | 2  | Create uncompressed NH5RDB database without input data.          |
|          | 3  | Create compressed NH5RDB database without input data.            |

Table 24 PARAMi Names and Descriptions

| Name     | Description, Type, and Default Values  |  |
|----------|--|--|
| H5GM34   | Write GEOM3 and GEOM4 data in NH5RDB   |  |
|          | 1  | Write GEOM3 and GEOM4 data in NH5RDB.  |
|          | 0  | Do not write GEOM3 and GEOM4 data in NH5RDB.   |
|          | -1   | Use OP2GM34 setting for GEOM3 and GEOM4 output (Default).                                |
| H5INFO   | Write job run information in NH5RDB or not.                                      |  |
|          | 1  | Yes (Default)  |
|          | 0  | No   |
| H5MDL    | Write model input data in separate file  |  |
|          | 1  | Write model input data into a separate file  |
|          | 0  | Do not write model input data into a separate file (Default)                             |
| H5MTX    | Write matrix data in separate file   |  |
|          | 1  | Write matrix data into a separate file   |
|          | 0  | Do not write matrix data into a separate file (Default)                                  |
| H5NORDOF | Parameter to suppress output rotational components to NH5RDB database            |  |
|          | 0  | Output both translational and rotational components to NH5RDB database (Default).        |
|          | 1  | Do not output rotational components to NH5DB database.<br>Only used when HDF5 is 0 or 1. |
|          |  |  |
| H5XHH    | Write BHH, MHH and KHH matrices in NH5RDB  |  |
|          | 0  | Do not write BHH, MHH and KHH in NH5RDB (Default)  |
|          | 1  | Write BHH, MHH and KHH in NH5RDB   |
| IGNSHBDN | Check adjacent shell elements have opposite direction of normal vector, if found |  |
|          | 0(default)   | Issue a user fatal message   |
|          | 1  | Issue a user warning message   |
| INTOUT   | Flag to control FORCE/STRESS/STRAIN OUTPUT location for QUADR/TRIAR elements     |  |
|          | 0  | Corner output. (Default)   |
|          | 1  | Integration point output.  |

Table 24 PARAMi Names and Descriptions

| Name                         | Description, Type, and Default Values   |   |
|------------------------------|---|---|
|                              | Integer   | Meaning   |
| LMT2MPC,<br>INTEGER<br>INPUT | Parameter to enable the use of elimination method to process Lagrange Multipliers.  |   |
|                              | 0   | Do not force the use of elimination method. (Default)   |
|                              | 1   | Use elimination method to process Lagrange Multipliers using unsymmetric method.  |
|                              | 2   | Use elimination method to process Lagrange Multipliers using symmetric method (may be faster).  |
| MLTSPLIN                     | Parameter to specify whether an aerodynamic grid can be splined more than once.   |   |
|                              | 0   | References on separate splines to the same aero grid are not allowed. (Default)   |
|                              | 1   | Aero grids can be referenced on multiple spline entries.  |
| MPCF129                      | Request the calculation of MPC forces in SOL129. The forces include only the contributions from linear elements.  |   |
|                              | 0   | Don not compute MPC Forces (default)  |
|                              | 1   | Compute MPC Forces  |
| NLDIFF                       | Flag to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis (parameter LGDISP=1) in SOL 400. Options 3, 4, and 5 are available only for elements with PSHNL1, PSHNL2, PSLN1, or PSHEARN Bulk Data entry. |   |
|                              | 0   | Compute. (Default except for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis)  |
|                              | 1   | Do not compute  |
|                              | 2   | The differential stiffness matrix and follower force stiffness will not be computed if the tangential stiffness matrix is negative definite (Default for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis). |
|                              | 3   | Include only the deviatoric part of the differential (or initial stress or geometric) stiffness   |
|                              | 4   | Include only the tensile part of the differential (or initial stress or geometric) stiffness  |
|                              | 5   | Include the stress at the beginning of the increment for the differential (or initial stress or geometric) stiffness  |

Table 24 PARAMi Names and Descriptions

| Name    | Description, Type, and Default Values  |  |
|---------|--|--|
| NONUPIV | Parameter to select the numeric compute kernel and pivoting methods in MSCLDL and MSCLU sparse direct solvers.   |  |
|         | 0  | Use the native Bunch-Kauffman threshold pivoting in MSCLDL, and the native threshold partial pivoting in MSCLU (Default).  |
|         | 1  | Use no numeric pivoting in MSCLDL and MSCLU. BLAS3 TRSMs are called to compute the pivot column update to improve performance. Ill-conditioned models may die of “singular matrix” during sparse factorization.  |
|         | 2  | Not documented.  |
|         | 3  | LAPACK SYTRFs with Bunch-Kaufman pivoting and GETRFs with partial pivoting are called to perform factorizations, and BLAS3 TRSMs are called to compute pivot column update to improve performance.   |
| NSGRDS4 | Number of structural grids to be used in dividing a SPLINE4 using the RIS spline method. The spline will be divided into NGRIDS/NSGRDS4 regions, where NGRIDS is the number of grids listed on the associated SET1 entry. (Integer > 0, default=0. If NGRIDS < NSGRDS4, or NSGRDS4 is not specified, no divisions will occur.) |  |
| OFFDEF  | Element offset definition. A flag to determine how shell elements and bar and beam elements behave when the user supplies ZOFF values on the shell connection entries (CQUAD4, CQUADR, CTRIA3, CTRIAR, CQUAD8, and CTRIA6) and WiA and WiB on CBAR, CBEAM, and CBEAM3 connection entries. (Character)                          |  |
|         | ELMOFF   | Standard Nastran offset method. The ZOFF rotate with the shell element. The WiA and WiB offsets for beams are fixed. MD Nastran R3 and earlier. (Character, Default)   |
|         | LROFF  | <b>Large rotation offsets.</b> The shell normal directions are used to define the offset direction at each shell grid for CQUAD4, CTRIA3, CQUADR, CTRIA4, CQUAD8 and CQUAD8 elements. For beams, WiA and Wib defines the offset direction. This method allows for thermal load effects on ZOFF for shells and WiA and WiB for beams. Thermal load effect for offset is computed based on the grid point or element temperature, and thermal coefficient of the element (see NOTHRM). The mass moment of inertia is computed for the offset due to the grid point location change introduced by offset. Differential stiffness is computed for the offset using the same method as that of the Lagrange formulation of the RBAR. Not supported in SOL106, SOL129 and SOL 200. (Character) |

Table 24 PARAMi Names and Descriptions

| Name   | Description, Type, and Default Values   |  |
|--------|---|--|
| NODIF  | LROFF is used but the differential stiffness effect is turned off.<br>(Character)   |  |
| NOTHRM | LROFF is used but the thermal load effects are turned off. The thermal load has two effects: 1) the location of thermal load changes due to offset and 2) the length of offset changes due to thermal load. Effect (1) is computed for all solution sequences and Effect (2) is computed for SOL 400 only. Both effects are turned off by NOTHRM. (Character) |  |
| NODT   | LROFF is used but the differential stiffness and thermal load effect are turned off. (Character)  |  |
| ELMZ   | LROFF is used but the element z-direction is used for the offset direction. IF PARAM, SNORM, 0.0 or the computed value for SNORM is greater than the PARAM,SNORM,value, then the LROFF option will revert to this method for CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8 and CTRIA6. This option is not applicable for beams. (Character)                          |  |
| NOMASS | LROFF is used but the no mass effects are included. (Character)   |  |
| NDMTZ  | LROFF is used but the element z-direction is used for the offset direction and the differential stiffness, the thermal load effects, and the mass effects are turned off. For CQUAD4 and CTRIA3 elements this method should get similar results to the standard ELMOFF method. (Character)  |  |
|        | <b>Note:</b> This entry only effects ZOFF calculations for ZOFF specified on the shell connection entries. For ZO specified on the PCOMP or PCOMPG entries, the standard ELMOFF method will be used.  |  |
|        | If the computed value for SNORM is greater than the PARAM,SNORM,value and the user wishes not to change the parameter value, the Bulk Data entry SNORM can be used to override the shell normal.  |  |
|        | Solution sequences affected: For linear - all solution sequences. For nonlinear - SOL 400 only. <i>The LROFF method is not implemented into SOLs 106 and 129.</i>   |  |
| PEXTS4 | Used in conjunction with NSGRDS4. After partitioning the spline, each of the smaller splines will be extended by PEXTS4 in each direction (top,bottom, left and right). The value is expressed in percent so that PEXTS4=10.0 would extend the four boundaries by 10%. Real (0.0<PEXTS4<100.0, default=10.0)  |  |

Table 24 PARAMi Names and Descriptions

| Name     | Description, Type, and Default Values  |   |
|----------|--|---|
| PIVTHRSH | Parameter to set the threshold ‘uu’ for numeric pivot selection in the MSCLU and Intel MKL Paradiso solvers.   |   |
|          | 0 (Default)  | uu is set to <ul style="list-style-type: none"> <li>■ MSCLU: <math>10^{-9}</math></li> <li>■ MKL Pardiso: <math>10^{-13}</math> for non-symmetric matrices and <math>10^{-8}</math> for symmetric indefinite matrices. For more general information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a>, iparm(10).</li> </ul> |
|          | -i   | uu is set to $10^{-i}$ for both MSCLU and MKL Pardiso (where i is between 1 and 20)   |
| PRDIDPVT | Control of pivoting in Intel MKL Pardiso, when solving symmetric indefinite matrices. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a> , iparm(21). |   |
|          | 0  | 1x1 diagonal pivoting   |
|          | 1  | 1x1 and 2x2 Bunch-Kaufman pivoting (Default)  |
|          | 2  | 1x1 diagonal pivoting without automatic iterative refinement  |
|          | 3  | 1x1 and 2x2 Bunch-Kaufman pivoting without automatic iterative refinement   |
| PRDITRFN | Control of iterative refinement step for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a> , iparm(8).                            |   |
|          | 0  | Solver automatically performs two steps of iterative refinement (Default)   |
|          | >0   | Maximum number of iterative refinement steps, up to a maximum of nine.  |
|          | <0   | Same as above, but residual is computed using extended precision. Maximum of nine steps.  |

Table 24 PARAMi Names and Descriptions

| Name     | Description, Type, and Default Values  |   |
|----------|--|---|
| PRDMTYPE | Control of matrix type for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Paridso documentation</a> , Input Parameters.                          |   |
|          | 1  | real structurally symmetric   |
|          | 2  | real symmetric positive definite (SOL 101 default)                  |
|          | -2   | real symmetric indefinite (SOL 400 default)                         |
|          | 3  | complex structurally symmetric                                      |
|          | 4  | complex Hermitian positive definite                                 |
|          | -4   | complex Hermitian indefinite  |
|          | 6  | complex symmetric (SOL 108 & 111 default)                           |
|          | 11   | real unsymmetric  |
|          | 13   | complex unsymmetric (SOL 107 default)                               |
| PRDOOC   | Control of out-of-core (OOC) solution algorithm in Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Paridso iparm Parameter Table</a> , iparm(60). |   |
|          | 0  | In-core mode  |
|          | 1  | Choose between in-core and OOC mode based on memory (opencore) size |
|          | 2  | Out-of-core mode  |
| PRDWMTCH | Control of weighted matching algorithm in Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Paridso iparm Parameter Table</a> , iparm(13).          |   |
|          | 0  | Disable matching (SOL 101 default)                                  |
|          | 1  | Enable matching (SOL 107, 108, 111, and SOL 400 default)            |
| PRTELAS  | Print list of ID's of CELASI elements that are deleted.  |   |
|          | NO<br>(or blank)   | Turn off the print. (Default)                                       |
|          | YES  | Turn on the print.  |
| PRTFAST  | Print list of ID's of CFAST elements that are deleted.   |   |
|          | NO<br>(or blank)   | Turn off the print. (Default)                                       |
|          | YES  | Turn on the print.  |
| PRTMASS  | Print list of ID's of CMASSi, CONM1 and CONM2 elements that are deleted.   |   |
|          | NO<br>(or blank)   | Turn off the print. (Default)                                       |
|          | YES  | Turn on the print.  |

Table 24 PARAMi Names and Descriptions

| Name    | Description, Type, and Default Values   |  |
|---------|---|--|
| PRTSEAM | Print list of ID's CSEAM elements that are deleted.   |  |
|         | NO<br>(or blank)  | Turn off the print. (Default)  |
|         | YES   | Turn on the print.   |
| PRTWELD | Print list of ID's of CWELD elements that are deleted.  |  |
|         | NO<br>(or blank)  | Turn off the print. (Default)  |
|         | YES   | Turn on the print.   |
| QR6ROT  | Parameter to determine whether the drilling degrees-of-freedom are to be deactivated for QUADR/TRIAR elements. If the drilling degrees-of-freedom are deactivated, the QUADR/TRIAR become elements similar to QUAD4/TRIA3. QR6ROT has the following values:   |  |
|         | 0   | The drilling degrees-of-freedom are active. (Default)  |
|         | 1   | The drilling degrees-of-freedom are deactivated for all QUADR/TRIAR element in the model.  |
|         | 2   | The drilling degrees-of-freedom are deactivated for those QUADR/TRIAR which have membrane stiffness only (MID2 and MID3 are blank on the PSHELL entry) |
| QRSHEAR | Parameter to select the off-plane shear formulation for the QUADR element. There are two types of off-plane shear formulations: the stiffness method and the flexibility method. The stiffness method is a new method implemented in QUADR. The flexibility method was the method implemented in the QUAD4 element. Therefore, if the flexibility method is selected, the solution results of QUADR are closer to those of QUAD4. QRSHEAR has the following values: |  |
|         | 0   | Use stiffness method if MID3 ≠ 0 on the PSHELL Bulk Data entry. Use the flexibility method if MID3 = 0. (Default)                                      |
|         | 1   | Use flexibility method.  |
|         | 2   | Use the stiffness method.  |

Table 24 PARAMi Names and Descriptions

| Name   | Description, Type, and Default Values   |  |
|--------|---|--|
| RDBOTH | Parameter to select Rayleigh damping approach for rotordynamics (compatibility with MSC Nastran 2005) implementation, Integer. A cumulative sum can be provided in case multiple features are desired in the analysis.  |  |
| 0      | Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)  |  |
| 1      | Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT is set to 0.0.   |  |
| 2      | Ignore circulation effects in rotodynamic analysis.   |  |
| 4      | Include effect of stress stiffening using method = 1 (see RFORCE entry)   |  |
| 8      | Include effect of stress stiffening using method = 2 (see RFORCE entry)   |  |
| 16     | This option treats structural damping for rotors using complex stiffness matrix for complex eigenvalue analysis. Here, circulation terms are ignored in the analysis.   |  |
| RELAXF | If there are SPRELAX entries, RELAXF=1 will result in the GI module outputting the GPGK datablock without relaxation while the GDGK datablock will include the relaxation effects.<br><br>=0 (Default) Relaxation is applied to all splines<br>=1 Relaxation is only applied to the splining of displacements.  |  |
| REUPSE | Choose method for handling rigid elements in superelements when RIGID=LAGRAN or LGELIM  |  |
| 0      | (Default) Automatic selection:<br><br>a. If ALPHA is zero for ALL r-elements in ALL superelements (SEID>0) then keep of these r-elements in the superelements with RIGID=LINEAR formulation.<br><br>b. If ALPHA is not zero for ANY r-element in ANY superelement (SEID>0) then transfer ALL r-elements in ALL superelements to the residual structure. |  |
| 1      | Transfer all r-elements in the superelements to the residual structure.   |  |

Table 24 PARAMi Names and Descriptions

| Name     | Description, Type, and Default Values  |   |
|----------|--|---|
| RMRBE3RT | Parameter to control if remove rotational dof from REFC of any RBE3 which reference grid point(REFG) is used by solid elements only and REFC has rotational dof..  |   |
|          | 0(default)   | Don't remove.   |
|          | 1  | Remove rotational dof.  |
| RSTIGNDP | A flag to determine if ignoring duplicate PARAM entries between a restart run and its initial run.   |   |
|          | 0(default)   | Don't ignore duplicating.   |
|          | >0   | Ignore duplicating.   |
| SHEARP   | Parameter to select the new HARDER shear panel over the classic GARVEY panel. The HARDER panel through testing appears to be superior to the GARVEY panel especially in cases of thermal loading. (Character-all required) |   |
|          | GARVEY   | GARVEY SHEAR PANEL (Default)  |
|          | HARDER   | HARDER SHEAR PANEL - NEW  |
| SHRTOQ4  | Flag to determine whether to convert a CSHEAR element to a CQUAD4 element for nonlinear analysis. Cannot be used with the PSHEARN entry.   |   |
|          | 0  | Do not convert CSHEAR to CQUAD4. (Default)  |
|          | 1  | Convert CSHEAR to CQUAD4.   |
|          | 2  | Do converting and print out converted Bulk Data entries on f06 file.                                |
| SPBLNDX  | Factor to be applied to D1 and D2 blend depths on the SPBLND1 and SPBLND2 bulk data entries for determining the structural grids in the blended region. (Real, $\geq 1.0$ , default = 1.0 )                                |   |
| STREQCNT | Flag to control calculation method for equivalent von Mises stress/strain at element center for advanced nonlinear solid elements in SOL 400.  |   |
|          | 0  | Average integration point equivalent stress/strain (Default).                                       |
|          | 1  | Average integration point stress/strain components then compute equivalent value at element center. |
| TWBRBML  | Parameter to select method for computing properties of PBARL/PBEAML.   |   |
|          | 0  | Select Finite Element Method. (Default)   |
|          | 1  | Select Beam Library Equations.  |

**MDMIAUX**

## Matrices from Marc - SOL 600

Specifies the DOMAINsolver command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only. (See the MDMIOUT Bulk Data entry.)

**Format:**

| 1       | 2      | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|--------|-------|---|---|---|---|---|---|----|
| MDMIAUX | ID     |       |   |   |   |   |   |   |    |
|         | “FMS”  | STRF1 |   |   |   |   |   |   |    |
|         |        | STRF2 |   |   |   |   |   |   |    |
|         | “EXEC” | STRE1 |   |   |   |   |   |   |    |
|         |        | STRE2 |   |   |   |   |   |   |    |

**Example:**

|         |      |  |  |  |  |  |  |  |  |
|---------|------|--|--|--|--|--|--|--|--|
| MDMIAUX | 2    |  |  |  |  |  |  |  |  |
|         | FMS  |  | ASSIGN OUTPUT2=MODEL1.OP2,UNIT=50,DELETE |  |  |  |  |  |  |
|         | EXEC |  | DOMAINsolver ACMS (PRINT=YES,UPFACT=3.5) |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | ID of a matching IAUX field on the corresponding MDMIOUT entry. (Integer; no Default)  |
| “FMS”      | Enter the string FMS to start a series of FMS strings that should appear before the SOL entry for the secondary spawned job. (Character; no Default) |
| STRFi      | Series of FMS strings, entry as many as desired. (Character; no Default)   |
| “EXEC”     | Enter the string EXEC to start a series of strings that should appear after the SOL entry for the secondary spawned job. (Character; no Default)     |
| STREi      | Series of Executive Control strings, enter as many as desired. (Character; no Default)   |

**Remarks:**

1. This entry is not active unless a matching MDMIOUT entry is found in the input.
2. STRFi and STREi may occupy fields 3-9. (columns 9 to 72).
3. Standard FMS and Executive Control continuation rules apply to the strings.

**MDMOUT**

Matrices from Marc - SOL 600

Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600. This entry may be used to generate External Superelements using DMIG Matrices or an MSC Adams MNF File from the Marc portion of a SOL 600 analysis. SOL 600 only. (See the MNF600 and DMIGOUT Bulk Data entries.)

**Format:**

| 1      | 2     | 3    | 4    | 5    | 6     | 7     | 8    | 9    | 10 |
|--------|-------|------|------|------|-------|-------|------|------|----|
| MDMOUT | ID    | IDOF | G1   | THRU | G2    | ITYPE | NAME | IAUX |    |
|        | IDOF2 | G3   | THRU | G4   | IDOF3 | G5    | THRU | G6   |    |
|        | etc.  |      |      |      |       |       |      |      |    |

**Example:**

|        |     |        |   |      |      |   |      |   |  |
|--------|-----|--------|---|------|------|---|------|---|--|
| MDMOUT | 100 | 123456 | 1 | THRU | 5456 | 1 | m343 | 1 |  |
|--------|-----|--------|---|------|------|---|------|---|--|

| Descriptor  | Meaning  |
|-------------|--|
| ID          | Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. (Integer; Default = 1)  |
| IDOF, IDOF2 | List of DOF's to be output (any or all of the integers 1-6 are acceptable). (Integer; Default = 123456)  |
| G1, G3      | Starting grid ID for reduced matrices. (Integer; Required; no Default)   |
| G2, G4      | Ending grid ID for reduced matrices. (Integer; Required; no Default)   |
| ITYPE       | Type of file to generate. (Integer; Default = 1) <ul style="list-style-type: none"> <li>1      DMIG Matrices</li> <li>2      Adams MNF</li> </ul>  |
| NAME        | Name of file containing the output (Character; no Default; limited to 8 characters, should be in lower case, <b>the extension .dmi is added automatically</b> )  |
| IAUX        | ID of a MDMIAUX entry if the secondary run requires the use of file management entries or additional Executive Control statements such as the DOMAININSOLVER option for parallel processing. (Integer; Default = 0) A value of zero means that no auxiliary entries are required.  |
| ISOL        | Solution sequence to run using the DMIG matrices. To speed up the solution, use DOMAININSOLVER ACMS (PARTOP=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112). (Integer absolute value > 100; Default = 0 which means do not run any solution sequence using the DMIG's created by Marc in this execution) |

Remarks:

1. The continuation line(s) are not required.
2. This entry corresponds to Marc's entry, SUPERELEM with a value of 1 in the second line 4th field and produces DMIG's or an MDF file for the initial geometry prior to any nonlinear iterations.
3. DMIG output will be in jid.marc\_dmigst\_0001.
4. The reduced matrices may be used in the Nastran analysis for eigenvalue extraction or any other purpose by invoking the CONTINUE=5 option on the SOL 600 entry.
5. If the SOL 600 CONTINUE options is invoked, case control commands and a bulk data entry include statements to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
6. ID must be 106 or 129 in the Executive Control statement, SOL 600.ID.
7. Only one MDMOUT entry should be entered per run. If more are entered, only the first will be used.
8. MNF controls for other solution sequences are ignored for SOL 600.
9. For the case where DMIG's are generated and a continuation option is used, the following Bulk Data parameters are usually required in addition to the MDMOUT entry:  
\$2345678x234567890123456x34567890123456  
param marcfile nastb.rc  
param marcfil1 m3356
10. For a more general form of the DMIG output, see Bulk Data entry, DMIGOUT.

## MDMIR1

### Module Reposition by Mirroring - Option 1

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

Format:

| 1      | 2    | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|------|----|----|----|---|---|---|---|----|
| MDMIR1 | MVID | P1 | P2 | P3 |   |   |   |   |    |

Example:

|        |     |    |    |      |  |  |  |  |  |
|--------|-----|----|----|------|--|--|--|--|--|
| MDMIR1 | 110 | 12 | 45 | 1125 |  |  |  |  |  |
|--------|-----|----|----|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MVID       | Move identification number to be referenced on an MDMOVE entry. (Integer>0).                 |
| Pi         | GRID or POINT identification numbers of three non-collinear points in Module 0. (Integer>0). |

Remarks:

1. MDMIR1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRIDs or POINTs referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDMIR2****Module Reposition by Mirroring - Option 2**

Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane in Module 0.

**Format:**

| 1      | 2    | 3   | 4   | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|------|-----|-----|------|---|---|---|---|----|
| MDMIR2 | MVID | CID | RID | AXES |   |   |   |   |    |

**Example:**

|        |     |  |    |    |  |  |  |  |  |
|--------|-----|--|----|----|--|--|--|--|--|
| MDMIR2 | 110 |  | 45 | XZ |  |  |  |  |  |
|--------|-----|--|----|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MVID       | Move identification number to be referenced on an MDMOVE entry. (Integer>0).  |
| CID        | Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be mirrored about CID's origin if RID is blank. (Integer $\geq 0$ , default=0)    |
| RID        | GRID or POINT identification number or coordinates of reference point through which mirror plane will be applied. If RID is blank, then the origin of CID will be used (Integer $\geq 0$ , default=0) |
| AXES       | Coordinate system axes pair: "XY", "XZ", "YZ", "YX", "ZX", or "ZY" (Character).   |

**Remarks:**

1. MDMIR2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDMOVE**

Defines a Module repositioning sequence

Defines a Module repositioning sequence by referencing MDMIRi, MDROTi, and MDTRAN entries.

**Format:**

| 1      | 2     | 3     | 4     | 5     | 6      | 7 | 8 | 9 | 10 |
|--------|-------|-------|-------|-------|--------|---|---|---|----|
| MDMOVE | MODID | MVID1 | MVID2 | MVID3 | -etc.- |   |   |   |    |

**Example:**

|        |    |    |    |    |  |  |  |  |  |
|--------|----|----|----|----|--|--|--|--|--|
| MDMOVE | 40 | 10 | 11 | 12 |  |  |  |  |  |
|--------|----|----|----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MODID      | Module identification number of a primary or copied module (Integer>0)                 |
| MVIDi      | Move identification number of an MDTRAN, MDROTi or MDMIRi Bulk Data entry (Integer>0). |

**Remarks:**

1. MDMOVE can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. In order to use MDMOVE, an MDBULK entry with the same MODID must be specified with TYPE="MOVE".
3. If MDMIRi is referenced on an MDMOVE then its MVID must be specified in the MVID1 field and not in the second or subsequent MVIDi fields.

**MDMPC****Multipoint Constraint Between Two or More Modules**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where  $u_j$  represents degree-of-freedom Cj at grid or scalar point Gj. Gj may be contained in different Modules.

Format:

| 1     | 2    | 3    | 4  | 5  | 6     | 7  | 8  | 9  | 10 |
|-------|------|------|----|----|-------|----|----|----|----|
| MDMPC | SID  | MOD1 | G1 | C1 | A1    |    |    |    |    |
|       | MOD2 | G2   | C2 | A2 | MOD3  | G3 | C3 | A3 |    |
|       | MOD4 | G4   | C4 | A4 | -etc- |    |    |    |    |

Example:

|       |    |    |    |      |     |   |   |       |  |
|-------|----|----|----|------|-----|---|---|-------|--|
| MDMPC | 3  | 10 | 28 | 3    | 6.2 |   |   |       |  |
|       | 11 | 2  |    | 4.29 | 21  | 1 | 4 | -2.91 |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| MODj       | Module IDs that contain the GRID entries for Gj. (Integer $\geq 0$ ).  |
| Gj         | Identification number of grid or scalar point. (Integer > 0)   |
| Cj         | Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.) |
| Aj         | Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)  |

Remarks:

1. The MDMPC entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the MPC entry description. All remarks and figures under MPC apply to MDMPC.

**MDMPLN****Module Reposition by Mirroring**

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

**Format:**

| 1      | 2     | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-------|----|----|----|---|---|---|---|----|
| MDMPLN | MODID | P1 | P2 | P3 |   |   |   |   |    |

**Example:**

|        |     |    |    |      |  |  |  |  |  |
|--------|-----|----|----|------|--|--|--|--|--|
| MDMPLN | 110 | 12 | 45 | 1125 |  |  |  |  |  |
|--------|-----|----|----|------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MODID      | Module identification number of a primary or secondary (copied) module (Integer>0).         |
| Pi         | GRID or POINT identification numbers of three non-collinear points in Module 0 (Integer>0). |

**Remarks:**

1. MDMPLN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MDMPLN=n command.
2. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDRBE2****Rigid Body Element Between Two or More Modules, Form 2**

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points. All grid points may be defined in different Modules.

**Format:**

| 1      | 2    | 3    | 4    | 5   | 6      | 7     | 8    | 9   | 10 |
|--------|------|------|------|-----|--------|-------|------|-----|----|
| MDRBE2 | EID  | MODN | GN   | CM  | MOD1   | GM1   | MOD2 | GM2 |    |
|        | MOD3 | GM3  | MOD4 | GM4 | MOD5   | GM5   | MOD6 | GM6 |    |
|        | MOD7 | GM7  | MOD8 | GM8 | -etc.- | ALPHA |      |     |    |

**Example:**

|        |     |     |     |    |     |    |       |    |  |
|--------|-----|-----|-----|----|-----|----|-------|----|--|
| MDRBE2 | 9   | 101 | 8   | 12 | 201 | 10 | 202   | 12 |  |
|        | 202 | 14  | 203 | 15 | 203 | 16 | 6.5-6 |    |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ ).   |
| MODN       | Module ID that contains the GRID entry for GN. ( $\text{Integer} \geq 0$ ).  |
| GN         | Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. ( $\text{Integer} > 0$ )   |
| CM         | Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. See Remark 12. under RBE2. (Integers 1 through 6 with no embedded blanks.) |
| MODi       | Module IDs that contain the GRID entries for GMj. ( $\text{Integer} \geq 0$ ).   |
| GMi        | Grid point identification numbers at which dependent degrees-of-freedom are assigned. ( $\text{Integer} > 0$ )   |
| ALPHA      | Thermal expansion coefficient. See Remark 11. under RBE2. (Real or blank)  |

**Remarks:**

1. The MDRBE2 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE2 entry description. All remarks and figures under RBE2 apply to MDRBE2.

**MDRBE3****Interpolation Constraint Element Between Two or More Modules**

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

**Format:**

| 1      | 2       | 3      | 4       | 5    | 6      | 7      | 8      | 9      | 10 |
|--------|---------|--------|---------|------|--------|--------|--------|--------|----|
| MDRBE3 | EID     | REFMOD | REFGRID | REFC | WT1    | C1     | MOD1,1 | G1,1   |    |
|        | MOD1,2  | G1,2   | MOD1,3  | G1,3 | WT2    | C2     | MOD2,1 | G2,1   |    |
|        | MOD2,2  | G2,2   | -etc.-  | WT3  | C3     | MOD3,1 | G3,1   | MOD3,2 |    |
|        | G3,2    | -etc.- | WT4     | C4   | MOD4,1 | G4,1   | MOD4,2 | G4,2   |    |
|        | -etc.-  |        |         |      |        |        |        |        |    |
|        | "UM"    | MOD1   | GM1     | CM1  | MOD2   | GM2    | CM2    |        |    |
|        |         | MOD3   | GM3     | CM3  | -etc.- |        |        |        |    |
|        | "ALPHA" | ALPHA  |         |      |        |        |        |        |    |

**Example:**

|        |       |       |     |      |     |     |     |    |  |
|--------|-------|-------|-----|------|-----|-----|-----|----|--|
| MDRBE3 | 14    | 1000  | 100 | 1234 | 1.0 | 123 | 101 | 1  |  |
|        | 102   | 3     | 103 | 5    | 4.7 | 1   | 101 | 2  |  |
|        | 102   | 4     | 103 | 6    | 5.2 | 2   | 101 | 7  |  |
|        | 102   | 8     | 103 | 9    | 5.1 | 1   | 104 | 15 |  |
|        | 105   | 16    |     |      |     |     |     |    |  |
|        | UM    | 101   | 100 | 14   | 102 | 5   | 3   |    |  |
|        |       | 103   | 7   | 2    |     |     |     |    |  |
|        | ALPHA | 6.5-6 |     |      |     |     |     |    |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)                             |
| REFMOD     | Module IDs that contain the GRID entry for REFGRID. (Integer ≥ 0).  |
| REFGRID    | Reference grid point identification number. (Integer > 0)   |
| REFC       | Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)                   |
| WTi        | Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)                                |
| Ci         | Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.) |
| MODi,j     | Module IDs that contain the GRID entries for Gi,j. (Integer ≥ 0).   |

|         |   |
|---------|---|
| Gi,j    | Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)   |
| "UM"    | Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character) |
| MODk    | Module IDs that contain the GRID entries for GMk. (Integer $\geq 0$ ).  |
| GMi     | Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)   |
| CMi     | Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)  |
| "ALPHA" | Indicates that the next number is the coefficient of thermal expansion. (Character)   |
| ALPHA   | Thermal expansion coefficient. See Remark 14. under RBE3. (Real or blank)   |

**Remarks:**

1. The MDRBE3 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE3 entry description. All remarks and figures under RBE3 apply to MDRBE3.

**MDRJNT****Rigid Joint Between Two Modules**

Defines a rigid joint element connecting two coinciding grid points in two different Modules.

**Format:**

| 1      | 2   | 3    | 4  | 5    | 6  | 7  | 8 | 9 | 10 |
|--------|-----|------|----|------|----|----|---|---|----|
| MDRJNT | EID | MODA | GA | MODB | GB | CB |   |   |    |

**Example:**

|        |   |    |   |    |   |       |  |   |  |
|--------|---|----|---|----|---|-------|--|---|--|
| MDRJNT | 5 | 11 | 1 | 21 | 2 | 12345 |  | 1 |  |
|--------|---|----|---|----|---|-------|--|---|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (Integer > 0)  |
| MODA,MODB  | Module IDs that contain the GRID entries for GA and GB in Modules MODA and MODB, respectively. (Integer $\geq 0$ ).   |
| GA, GB     | Grid point identification numbers. (Integer > 0)  |
| CB         | Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. under RJOINT. (Integers 1 through 6 with no embedded or blank.) |

**Remarks:**

1. The MDRJNT entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RJOINT entry description. All remarks and figures under RJOINT apply to MDRJNT.

**MDROT1****Module Reposition by Rotation - Option 1**

Defines a Module rotation by specifying a rotation vector and reference point in Module 0.

**Format:**

| 1      | 2    | 3   | 4   | 5   | 6   | 7 | 8 | 9 | 10 |
|--------|------|-----|-----|-----|-----|---|---|---|----|
| MDROT1 | MVID | RID | IDS | IDE | MAG |   |   |   |    |

**Example:**

|        |     |  |    |    |     |  |  |  |  |
|--------|-----|--|----|----|-----|--|--|--|--|
| MDROT1 | 110 |  | 45 | 72 | 90. |  |  |  |  |
|--------|-----|--|----|----|-----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MVID       | Move identification number to be referenced on an MDMOVE entry. (Integer>0).  |
| RID        | GRID or POINT identification number of a reference point at which rotation will occur. If blank or 0 then Module will be rotated about rotation vector defined from IDS to IDE. (Integer $\geq 0$ , Default=0). |
| IDS        | GRID or POINT identification number of rotation vector's starting point. (Integer>0)  |
| IDE        | GRID or POINT identification number of rotation vector's end point. (Integer>0)   |
| MAG        | Magnitude of rotation in degrees. (Real $\neq 0.0$ ).   |

**Remarks:**

1. MDROT1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDROT1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDROT2****Module Reposition by Rotation - Option 2**

Defines a Module rotation by specifying a coordinate system axis for the rotation vector.

**Format:**

| 1      | 2    | 3   | 4    | 5    | 6   | 7 | 8 | 9 | 10 |
|--------|------|-----|------|------|-----|---|---|---|----|
| MDROT2 | MVID | CID | RIDS | AXIS | MAG |   |   |   |    |

**Example:**

|        |    |  |    |   |     |  |  |  |  |
|--------|----|--|----|---|-----|--|--|--|--|
| MDROT2 | 40 |  | 45 | Y | 90. |  |  |  |  |
|--------|----|--|----|---|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| MVID       | Move identification number to be referenced on an MDMOVE entry. (Integer>0).   |
| CID        | Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be rotated about CID's origin if RID is blank. (Integer $\geq 0$ , default=0). |
| RID        | GRID or POINT identification number of a reference point at which rotation will occur. If RID is blank or 0 then the origin of CID will be used. (Integer $\geq 0$ , Default=0)                    |
| AXIS       | Coordinate system axis: "X", "Y", "Z", "MX", "MY", or "MZ". The latter three values indicate the negative side of the axis. (Character)  |
| MAG        | Magnitude of rotation in degrees. (Real $\neq 0.0$ ).  |

**Remarks:**

1. MDROT2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDROT2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDRROD****Rigid Pin-Ended Element Connection Between Two Modules**

Defines a pin-ended element that is rigid in translation between two Modules.

**Format:**

| 1      | 2   | 3    | 4  | 5    | 6  | 7   | 8   | 9     | 10 |
|--------|-----|------|----|------|----|-----|-----|-------|----|
| MDRROD | EID | MODA | GA | MODB | GB | CMA | CMB | ALPHA |    |

**Example:**

|   |        |    |    |   |    |   |   |       |  |
|---|--------|----|----|---|----|---|---|-------|--|
| * | MDRROD | 14 | 11 | 1 | 21 | 2 | 2 | 6.5-6 |  |
|---|--------|----|----|---|----|---|---|-------|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| MODA,MODB  | Module IDs that contain the GRID entries for GA and GB (Integer $\geq 0$ ).  |
| GA, GB     | Grid point identification numbers of connection points in Modules MODA and MODB, respectively. (Integer $> 0$ )  |
| CMA,CMB    | Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. See Remark 3. under RROD. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear MDRROD. For Lagrange MDRROD, both CMA and CMB can be blank.) |
| ALPHA      | Thermal expansion coefficient. See Remark 11. under RROD. (Real or blank)  |

**Remarks:**

1. The MDRROD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RROD entry description. All remarks and figures under RROD apply to MDRROD.

**MDSEAM****A Shell Patch SEAM Connection Between Two Modules**

Defines a SEAM connecting two surface patches in two Modules.

**Format:**

| 1      | 2   | 3   | 4    | 5     | 6    | 7    | 8    | 9    | 10 |
|--------|-----|-----|------|-------|------|------|------|------|----|
| MDSEAM | EID | PID | SMLN | CTYPE | IDAS | IDBS | IDAE | IDBE |    |
|        | GS  | GE  |      |       |      |      | MODA | MODB |    |

**Alternate Format:**

| 1      | 2   | 3   | 4  | 5     | 6    | 7    | 8    | 9    | 10 |
|--------|-----|-----|----|-------|------|------|------|------|----|
| MDSEAM | EID | PID |    | CTYPE | IDAS | IDBS | IDAE | IDBE |    |
|        | XS  | YS  | ZS | XE    | YE   | ZE   | MODA | MODB |    |

**Example:**

|        |       |       |  |  |    |    |    |    |  |
|--------|-------|-------|--|--|----|----|----|----|--|
| MDSEAM | 552   | 297   |  |  | 43 | 48 |    |    |  |
|        | 30422 | 77987 |  |  |    |    | 15 | 32 |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| PID        | Property identification number of a PSEAM entry. (Integer > 0)  |
| SMLN       | SEAM line identification. See Remark 2. under CSEAM. (CHAR or blank)  |
| CTYPE      | Connectivity search type. (Character)   |
|            | If CTYPE = “PSHELL”, IDAS and IDBS are property identification numbers of PSHELL’s. (Default)   |
|            | If CTYPE = “ELEM”, IDAS and IDBS are element identification numbers.  |
| IDAS, IDBS | Used to define patch A and B in Modules MODA and MODB or the start of patch A or B for a tailored blank in Modules MODA and MODB. See Remark 2. under CSEAM (Integer > 0)   |
|            | If CTYPE = “PSHELL”, required property id defining patches A and B in Modules MODA and MODB. If CTYPE = “PSHELL” and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom. See Remark 6. under CSEAM. |
|            | If CTYPE = “ELEM”, required element id defining patches A and B in Modules MODA and MODB. IDAS ≠ IDBS.  |

| Descriptor | Meaning  |
|------------|--|
| IDAE, IDBE | Used to define the end of patch A and the end of patch B for a tailored blank in Modules MODA and MODB. See Remark 4. under CSEAM. (Integer > 0 or blank)<br><br>If CTYPE = "PSHELL", property id defining patches A and B. If CTYPE = 'PSHELL' and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom.<br><br>If CTYPE = "ELEM", element id defining patches A and B in Modules MODA and MODB. IDAE ≠ IDBE. |
| GS, GE     | Grid ids of piercing points on patches A and B of the Start and End of the SEAM. Their GRID entries must be defined in the main Bulk Data section (Module 0). (Integer > 0)  |
| XS,YS,ZS   | Location of the SEAM Start. (Real or blank)  |
| XE,YE,ZE   | Location of the SEAM End. (Real or blank)  |
| MODA,MODB  | Module IDs that contain the surface patches defined by property or element entries IDAS/IDAE and IDBS/IDBE, respectively (Integer ≥ 0).  |

#### Remarks:

1. The MDSEAM entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CSEAM entry description. All remarks and figures under CSEAM apply to MDSEAM.

**MDTRAN****Module Reposition by Translation**

Defines a Module translation by specifying a vector in Module 0.

**Format:**

| 1      | 2    | 3   | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|------|-----|-----|-----|---|---|---|---|----|
| MDTRAN | MVID | IDS | IDE | MAG |   |   |   |   |    |

**Example:**

|        |     |    |    |  |  |  |  |  |  |
|--------|-----|----|----|--|--|--|--|--|--|
| MDTRAN | 110 | 72 | 45 |  |  |  |  |  |  |
|--------|-----|----|----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| MVID       | Move identification number to be referenced on an MDMOVE entry. (Integer>0).                                |
| IDS        | GRID or POINT identification number of translation vector's starting point. (Integer>0).                    |
| IDE        | GRID or POINT identification number of translation vector's end point. (Integer>0).                         |
| MAG        | Magnitude of translation. If blank or zero, then the length of the vector will be used. (Real, Default=0.0) |

**Remarks:**

1. MDTRAN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDTRAN must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDWELD****Weld or Fastener Element Connection Between Two Modules**

Defines a weld or fastener connecting two surface patches or points between two Modules.

**Format PARTPAT:**

| 1      | 2    | 3    | 4  | 5         | 6    | 7    | 8 | 9    | 10 |
|--------|------|------|----|-----------|------|------|---|------|----|
| MDWELD | EWID | PWID | GS | "PARTPAT" | GA   | GB   |   | MCID |    |
|        | PIDA | PIDB |    |           | MODA | MODB |   |      |    |
|        | XS   | YS   | ZS |           |      |      |   |      |    |

**Example:**

|        |     |    |     |         |    |    |  |  |  |
|--------|-----|----|-----|---------|----|----|--|--|--|
| MDWELD | 101 | 8  | 203 | PARTPAT |    |    |  |  |  |
|        | 21  | 33 |     |         | 11 | 21 |  |  |  |

Alternate formats and examples:

**Format ELPAT:**

| 1      | 2     | 3     | 4  | 5       | 6    | 7    | 8 | 9    | 10 |
|--------|-------|-------|----|---------|------|------|---|------|----|
| MDWELD | EWID  | PWID  | GS | "ELPAT" | GA   | GB   |   | MCID |    |
|        | SHIDA | SHIDB |    |         | MODA | MODB |   |      |    |
|        | XS    | YS    | ZS |         |      |      |   |      |    |

**Example:**

|        |     |     |     |       |    |    |  |  |  |
|--------|-----|-----|-----|-------|----|----|--|--|--|
| MDWELD | 103 | 5   | 403 | ELPAT |    |    |  |  |  |
|        | 309 | 511 |     |       | 11 | 21 |  |  |  |

**Format ELEMID:**

|        |       |       |      |          |      |      |  |      |  |
|--------|-------|-------|------|----------|------|------|--|------|--|
| MDWELD | EWID  | PWID  | GS   | "ELEMID" | GA   | GB   |  | MCID |  |
|        | SHIDA | SHIDB | MODS |          | MODA | MODB |  |      |  |

**Example:**

|        |     |     |     |        |    |    |  |  |  |
|--------|-----|-----|-----|--------|----|----|--|--|--|
| MDWELD | 103 | 5   | 403 | ELEMID |    |    |  |  |  |
|        | 309 | 511 |     |        | 11 | 12 |  |  |  |

## Format GRIDID:

|        |      |      |     |          |      |      |       |      |  |
|--------|------|------|-----|----------|------|------|-------|------|--|
| MDWELD | EWID | PWID | GS  | "GRIDID" | GA   | GB   | SPTYP | MCID |  |
|        |      |      |     |          | MODA | MODB |       |      |  |
|        | GA1  | GA2  | GA3 | GA4      | GA5  | GA6  | GA7   | GA8  |  |
|        | GB1  | GB2  | GB3 | GB4      | GB5  | GB6  | GB7   | GB8  |  |

## Example:

|        |    |    |     |        |    |    |    |    |  |
|--------|----|----|-----|--------|----|----|----|----|--|
| MDWELD | 7  | 29 | 233 | GRIDID |    |    | QT |    |  |
|        |    |    |     |        | 11 | 21 |    |    |  |
|        | 15 | 28 | 31  | 35     | 46 | 51 | 55 | 60 |  |
|        | 3  | 5  | 8   |        |    |    |    |    |  |

## Format ALIGN:

|        |      |      |  |         |      |      |  |      |  |
|--------|------|------|--|---------|------|------|--|------|--|
| MDWELD | EWID | PWID |  | "ALIGN" | GA   | GB   |  | MCID |  |
|        |      |      |  |         | MODA | MODB |  |      |  |

## Example:

|        |   |    |  |       |     |     |  |  |  |
|--------|---|----|--|-------|-----|-----|--|--|--|
| MDWELD | 7 | 29 |  | ALIGN | 103 | 259 |  |  |  |
|        |   |    |  |       | 11  | 21  |  |  |  |

| Descriptor | Meaning  | Type                      | Default  |
|------------|--|---------------------------|----------|
| EWID       | MDWELD element identification number.<br>See Remark 1 under CWELD.   | 0 < Integer < 100,000,000 | Required |
| PWID       | Property identification number of a PWELD entry.   | Integer > 0               | Required |
| GS         | Identification number of a grid point which defines the location of the connector. GRID entry for GS must be defined in main Bulk Data section only (Module 0) except for a point-to-patch connection with OPTION="ELEMID" in which case GS must be defined in Module MODS. See Remarks 2. and 3. under CWELD. | Integer > 0 or blank      |          |

| Descriptor | Meaning  | Type                  | Default                                |
|------------|--|-----------------------|--|
|            | Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA in Module MODA and PIDB in Module MODB, respectively. The "PARTPAT" format connects up to 3x3 elements per patch. See Remark 4. under CWELD. | Character             | Required                               |
| GA, GB     | Grid point identification numbers of piercing points on surface A and surface B, respectively. GRID entries for GA and GB must be defined in the main Bulk Data section only (Module 0) except for OPTION="ALIGN". See Remark 5. under CWELD.  | Integer > 0 or blank  | Blank                                  |
| MCID       | Specifies the element stiffness coordinate system. See Remark 16. under CWELD  | Integer > -1 or blank | Default = -1                           |
| MODS       | Module ID that contains GRID entry for GS. Only used for point-to-patch connection with OPTION='ELEMID'.   | Integer ≥ 0           | Default = 0                            |
| MODA, MODB | Module IDs that contain the surface patches defined by (1) property entries PIDA and PIDB, (2) element entries SHIDA and SHIDB or (3) GRID entries GAI and GBi, respectively.  | Integer ≥ 0           | Default = 0                            |
| PIDA, PIDB | Property identification numbers of PSHELL entries defining surface A in Module MODA and B in Module MODB, respectively.  | Integer > 0           | Required for "PARTPAT"                 |
| XS, YS, ZS | Coordinates of spot weld location in basic. See Remark 2. under CWELD.   | Real                  | Required if GS and GA are not defined. |

For the alternate formats, the descriptor meaning are described below:

| Descriptor   | Meaning   | Type        | Default               |
|--------------|---|-------------|-----------------------|
| "ELPAT"      | Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The "ELPAT" format connects up to 3x3 elements per patch. See Remark 6. under CWELD.                                 | Character   | Required              |
| SHIDA, SHIDB | Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.  | Integer > 0 | Required for "ELPAT"  |
| "ELEMID"     | Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The "ELEMID" format connects one shell element per patch. See Remark 7. under CWELD.                                 | Character   | Required              |
| SHIDA, SHIDB | Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.  | Integer > 0 | Required for "ELEMID" |
| "GRIDID"     | Character string indicating that the connectivity of surface patch A in Module MODA to surface patch B in Module MODB is defined with two sequences of grid point identification numbers, GAi in Module MODA and GBi in Module B, respectively. The "GRIDID" format connects the surface of any element. See Remark 8. under CWELD. | Character   | Required              |
| SPTYP        | Character string indicating types of surface patches A and B. SPTYP = "QQ", "TT", "QT", "TQ", "Q" or "T". See Remark 9. under CWELD.  | Character   | Required for "GRIDID" |

| Descriptor | Meaning   | Type        | Default               |
|------------|---|-------------|-----------------------|
| GAi        | Grid identification numbers of surface patch A. GA1 to GA3 are required. Their GRID entries must be defined in Module MODA only. See Remark 10. under CWELD.  | Integer > 0 | Required for "GRIDID" |
| GBi        | Grid identification numbers of surface patch B. Their GRID entries must be defined Module MODB only. See Remark 10. under CWELD.  | Integer > 0 |                       |
| "ALIGN"    | Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA in Module MODA and GB in Module MODB, respectively. See Remark 11. under CWELD. | Character   | Required              |
| GA, GB     | Vertex grid identification number of shell A and B, respectively. GA and GB GRID entries must be defined in Module MODA and Module MODB, respectively.  | Integer > 0 | Required for "ALIGN"  |

#### Remarks:

1. The MDWELD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CWELD entry description. All remarks and figures under CWELD apply to MDWELD.

**MESH**

## Mesh Generator

Defines a mesh. Used in SOL 700 only.

Format:

| 1    | 2       | 3     | 4     | 5       | 6       | 7     | 8     | 9    | 10 |
|------|---------|-------|-------|---------|---------|-------|-------|------|----|
| MESH | MID     | TYPE  | DXEL  | DYEL    | DZEL    | XREF  | YREF  | ZREF |    |
|      | X0      | Y0    | Z0    | DX      | DY      | DZ    |       |      |    |
|      | NX      | NY    | NZ    | SUBMESH | NSTGP   | NSTEL | PROP  | PID  |    |
|      | RESIZE  | TID-X | TID-Y | TID-Z   | METHOD  |       |       |      |    |
|      | BIAS    | GROWX | GROWY | GROWZ   | IBIDX   | IBIDY | IBIDZ |      |    |
|      | XOBX    | YOBX  | ZOBY  | DXBX    | DYBX    | DZBX  |       |      |    |
|      | CID     |       |       |         |         |       |       |      |    |
|      | NELCUBE | NBX   | NBY   | NBZ     | PROCDIR | NPX   | NPY   | NPZ  |    |

Example:

| Data |        |       |     |     |     |  |  |  |  |
|------|--------|-------|-----|-----|-----|--|--|--|--|
| MESH | 1      | ADAPT | 0.1 | 0.2 | 0.3 |  |  |  |  |
|      |        |       |     |     |     |  |  |  |  |
|      |        |       |     |     |     |  |  |  |  |
|      | SCALE  | 101   |     |     | ALL |  |  |  |  |
|      | CENTER | 1.2   | 1.2 | 1.2 |     |  |  |  |  |
|      |        |       |     |     |     |  |  |  |  |

| Descriptor         | Meaning  |
|--------------------|--|
| MID                | Unique MESH number. See Remark 1. (Integer > 0; no Default)  |
| TYPE               | Type of mesh generation: See Remark 1. (Character; Required)   |
| ADAPT              | An Euler mesh will be created around a coupling surface. This option is only valid for PROP=EULER, and requires that the MID of the MESH is referenced from the MESHID of a COUPLE card. During the simulation, when the coupling surface moves or deforms, the Euler mesh will adapt itself by adding and removing elements. The adapt algorithm ensures that the coupling surface is contained inside the Euler mesh at all times with the minimum amount of elements. The Euler elements are aligned with the basic coordinate. |
| BOX                | A rectangular mesh will be created, that is aligned with the basic coordinate system. The mesh will use CHEXA elements.  |
| DXEL,DYEL,<br>DZEL | Euler element sizes. See Remark 1. (Real)  |

| Descriptor     | Meaning   |
|----------------|---|
| XREF,YREF,ZREF | Coordinates of reference point.<br>For TYPE=ADAPT, these coordinates provide control over the location of the Euler mesh, to avoid that faces of the Euler mesh are initially at the same location as faces of the coupling surface. For TYPE=BOX, these coordinates will be used as the origin of the mesh. They are the default setting for (X0, Y0, Z0). (Real; Default = -1e-6) |
| X0,Y0,Z0       | Coordinates of point of origin. (Real, XREF, YREF, ZREF)<br>Not used for TYPE=ADAPT   |
| DX,DY,DZ       | Width of mesh in different directions.<br>For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Real)  |
| NX,NY,NZ       | Number of elements in the different directions.<br>For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Integer > 0)  |
| SUBMESH        | Allows using smaller mesh sizes for a part of the mesh. SUBMESH is the MESH ID of a finer mesh that is to replace part of the mesh. See Remark 8. (Integer $\geq 0$ , default=0)  |
| NSTGP          | Starting grid-point number.<br>Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting grid-point number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)  |
| NSTEL          | Starting element number.<br>Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting element number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)  |
| PROP           | Property type: (Character; default=EULER)<br>EULER An Eulerian mesh will be created.  |
| PID            | ID of a PEULER or PEULER1 entry. (Integer > 0, Required.)   |
| RESIZE         | Only valid for TYPE=ADAPT. See Remark 6. (Character, NONE)<br>Option to change the element size during the simulation:<br>NONE No resizing of DX,DY,DZ during simulation.<br>SCALE The elements are resized by a scale-factor as a function of time.<br>LENGTH The elements are resized by specifying the length as a function of time.   |
| TID-X          | ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; Blank)   |
| TID-Y          | ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)   |
| YID-Z          | ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)   |
| METHOD         | Method for determining when to create Euler elements: (Character; ALL)  |

| Descriptor                            | Meaning  |
|---------------------------------------|--|
| ALL                                   | Always re-mesh any existing Euler element. Maintains existing void regions. Only used for resizing.  |
| MATERIAL                              | Only re-mesh those Euler elements that contain material. Removes void regions. See Remark 7.   |
| BIAS                                  | Adds bias to the mesh. (Character; Blank)  |
| CENTER                                | Starting at the center of the BOX the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center.   |
| REF                                   | Starting at the reference point the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center  |
| GROWX,<br>GROWY,<br>GROWZ             | Total grow factor. Is the ratio between finest and coarsest element size. (Real > 0; Only required if BIAS is not blank)   |
| IBIDX,<br>IBIDY,<br>IBIDZ             | BIAS ID reference. See Remarks 9. and 10. (Integer; Default = 0)   |
| X0BX,Y0BX,Z<br>0BX,DXBX,D<br>YBX,DZBX | Definition of an auxiliary box for output purposes. By defining an auxiliary box all adaptive elements that are within the box for one of the cycles requested are stored in the archive. This allows multiple cycles in one Euler archive. This box should be sufficiently large such that it contains all elements. The fields X0BX, Y0BX and Z0BX specify the start point and DXBX, DYBX and DZBX specify the width of box. If the there are adaptive elements outside the box the run is terminated and a larger box needs to be specified. Only used for TYPE=ADAPT. (Real; Blank) See Remark 14. |
| CID                                   | ID - number of a local coordinate system. (Integer $\geq 0$ ; Default=0)   |
| NELCUBE                               | The number of elements per cube. This number is used as a guideline. The actual number used per cube can differ and can be found in the OUT file. See Remark 12. (Integer $\geq 1$ ; Default=1)  |
| NBX                                   | Overrules NELCUBE. The number of cubes in the x-direction. (Integer $\geq 0$ ; Default=1)  |
| NBY                                   | The number of cubes in the y-direction.(Integer $\geq 0$ ; Default=NBX)  |
| NBZ                                   | The number of cubes in the z-direction. (Integer $\geq 0$ ; Default=NBX)   |
| PROCDIR                               | This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Char, Default=X)<br><br>X Partition in global X direction first.<br>Y Partition in global Y direction first.<br>Z Partition in global Z direction first.<br>USER Define user defined partitioning.   |

| Descriptor | Meaning   |
|------------|---|
|            | SIMPLE Partition Euler cubes in a simple pattern.                                       |
| NPX        | The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer, Default=1) |
| NPY        | The number of cubes in the y-direction. (Integer, Default=NPX)                          |
| NPZ        | The number of cubes in the z-direction. (Integer, Default=NPX)                          |

**Remarks:**

1. The grid-points of the mesh are generated at following locations:

Type=ADAPT:  $(x,y,z) = (XREF+i^*DXEL, YREF+j^*DYEL, ZREF+k^*DZEL)$

Grid-points and elements located a certain distance outside the coupling surface will not be created. This saves memory and CPU time.

When  $(XREF, YREF, ZREF)$  are outside the coupling surface, no grid-point will be created at this location, but the mesh will be shifted appropriately.

Type=BOX:  $(x,y,z) = (X0+i^*DXEL, Y0+j^*DYEL, Z0+k^*DZEL)$

Nodes and Elements will always be created, even if the MESH is referenced from the MESHID of a COUPLE entry.

One of the following input combinations is required:

TYPE=ADAPT

a. (DXEL, DYEL, DZEL)

or

b. 1

c. (DX, DY, DZ) and (NX, NY, NZ)

→ DXEL = DX/NX ; DYEL=DY/NY ; DZEL=DZ/NZ

TYPE=BOX

a. (DXEL, DYEL, DZEL) and (NX, NY, NZ)

or

b. (DX, DY, DZ) and (NX, NY, NZ)

→ DXEL = DX/NX ; DYEL=DY/NY ; DZEL=DZ/NZ

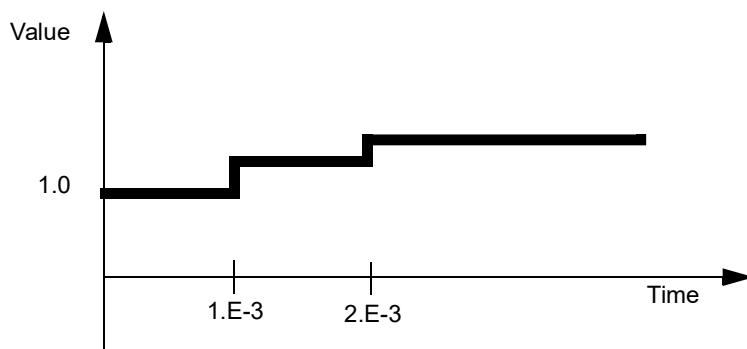
2. When the starting grid point and/or element number is left blank, then the default start number for the elements and grid-points is equal to the maximum number used +1. For simulations with multiple coupling surfaces two methods of treating transport between the Euler meshes are available. One method supports meshes of TYPE = ADAPT, but does not allow the specification of starting element or starting grid-point number.

3. The PID should refer to an existing property id, which can handle the property type given by PROP.

4. To avoid that the Euler mesh will be resized every time-step, the functions defined by TID-X, TID-Y, TID-Z must describe a ‘step-function’, like in this example:

- i. TABLED1,1,,,,,+
- ii. +,0.0,1.0,,,,,+
- iii. +,1.E-3,1.0,,,,,+
- iv. +,1.E-3,1.1,,,,,+
- v. +,2.E-3,1.1,,,,,+
- vi. +,2.E-3,1.2

Which specifies following function:



5. Care must be taken when refining the Euler mesh. To avoid instabilities, it is advised to stay within the following guidelines:
  - a. Each refining step, use a scale factor larger than 0.5
  - b. Allow the solution to become smooth again after each refining step. For airbag simulations, use an interval larger than  $5 * \text{diameter\_airbag} / \text{soundspeed}$
6. Resizing is not available for the Multi-material solver.
7. In most cases METHOD = ALL is the preferred method. Using METHOD=MATERIAL may be helpful in case of instabilities due to presence of void regions.
8. SUBMESH glues a fine mesh into a coarse mesh and uses the same gluing functionality as PARAM, GRADEDMESH. If the fine mesh is completely contained inside the coarse mesh no restrictions apply. Then to avoid any restrictions the grid points of the fine mesh are slightly displaced. But if parts of the fine mesh are outside the coarse mesh a restriction applies. In that case an Euler element of the coarse mesh has to be fully active or fully inactive. This means that the coarse element should not intersect elements of the fine mesh or it should be fully covered by the fine elements. Fine elements are not allowed to cover any part of the coarse elements. In practice, this means that the fine mesh has to fit nicely in the coarse mesh. For details refer to the section "Graded meshes" in the MSC Nastran Explicit Nonlinear User's Manual.

When running on one cpu the elements of the Euler mesh and the submesh will be put into one euler archive. But when running with multiple cpus, the mesh and sub mesh will be put in different euler archives. To distinguish the archives, the name \_FVX is added to the Euler archive names. Here X is the MESH-ID MID. The Euler archive of the mesh and the submesh can be read simultaneously into Patran.

9. A biased mesh has nonconstant element sizes in selected directories. Neighbor element size can have a constant ratio or have identical size. In literature, this type of mesh is also referred to as a nonuniform mesh or a locally refined mesh.
10. A block mesh consists of a number of planes in all three directions. For a nonbiased mesh, these planes are at fixed distance from each other. In a biased mesh, the distance between subsequent planes can differ. The varying element size is determined by:
  - a. IBIDX
  - b. GROWX
  - c. The constant step size specified by X0, NX, DX.

Here, IBIDX overrules GROWX and GROWX overrules the X0, NX DX specification. Likewise, for the other directions. The locations of the planes are written out in the OUT file. Intersecting an x-plane with a y-plane and z-plane will give a grid point. By carrying out all intersections, the grid points are constructed.

11. Defining the CID allows for positioning the mesh box arbitrarily in space. When active, the following restrictions apply:
  - MESH,ADAPT cannot be used
  - The use of single material Euler with strength elements is not supported

12. Setting NELCUBE, NBX, NBY, NBZ allow to divide the Euler mesh into a number of cubes. By setting NELCUBE equal to 2000, optimal use is made of memory caching during Euler computation. This can give a speedup of 1.5. If NBX is defined, also NBY and NBZ need to be defined. Defining NBX overrules the definition of NELCUBE.

When using multiple Euler cubes the BARRIER and FLOW definitions only support geometric conditions like boundary face direction and square definition.

13. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.

When option PROCDIR=USER, the values for NBX, NBY, and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option, NPX\*NPY\*NPZ must be equal to the number of processors used in the cluster.

For option PROCDIR=SIMPLE, the values NBX, NBY, and NBZ on DYPARAM,EULERCB must be such that NBX\*NBY\*NBZ is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4, NBX\*NBY\*NBZ must be equal to either 4 or 8 or 12, etc. Otherwise, the calculation will terminate prematurely with an error message.

When using these options the Barrier and FLOW definitions only support geometric conditions like boundary face direction and a square definition

14. To determine a suitable size for the static output box, the simulation can be run first without the fields X0BX, Y0BX, Z0BX, DXBX, DYBX, and DZBX set. In the ,OUT file each summary of adaptive meshing gives the smallest box surrounding all adaptive elements so far. The last summary then yields the dimension of the static output box. To accommodate for elements that are not completely inside the box, the actual static output box is automatically extended a little. The actual dimensions are written in the out file after the first adaptive meshing summary. Is not needed to set the XREF, YREF, and ZREF option. If they are set the static output box will be compatible with the defined reference point.

**MESUPER**

Defines External Superelement DMIG Input for SOL 600 Residual Analyses

Superelement DMIG matrices are created by Nastran and used when Marc is spawned from Nastran, SOL 600 only.

**Format:**

| 1       | 2  | 3 | 4 | 5 | 6     | 7 | 8 | 9 | 10 |
|---------|----|---|---|---|-------|---|---|---|----|
| MESUPER | ID |   |   |   | Fname |   |   |   |    |

**Example:**

|         |   |            |  |
|---------|---|------------|--|
| MESUPER | 1 | super1.pch |  |
| MESUPER | 2 | super2.pch |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Superelement ID. (Integer; no Default)  |
| Fname      | Filename containing external superelement data from the creation run (using Case Control EXTSEOUT (ASMBULK, DMIGPCH,EXTID=ID) (Character; no Default). Left justify in field, see Remark 8. |

**Remarks:**

1. Enter as many MESUPER lines as necessary to define all external superelements.
2. This entry can presently only be used with SOL 600,106, SOL 600,101, SOL 600,nlstatic, or SOL 600,sestatic. External Superelements are not presently available for other types of SOL 600 analyses such as nonlinear (or linear) transient dynamics, eigenvalue analysis or buckling.
3. Fname is limited to 56 characters.
4. Include entries with the same Fname must be specified as include files in the Nastran input file. The include specifications must appear at the end of the Bulk Data portion of the file.
5. External superelement creation runs should use the Case Control command:  
**EXTSEOUT(ASMBULK,DMIGPCH,EXTID=N)**  
where N is the external superelement ID number. All creation runs must have the same number of subcases and use the same subcase IDs.
6. The SOL 600 residual input file must have the same number of subcases and subcase numbers as the creation runs.
7. This entry acts like an element, in other words it is not controlled by a case control command. It is always active if entered.

8. The MESUPER entries should normally be coded in small fixed field format. If coded in small format free field, Fname is limited to 8 characters. If coded in large format free field, Fname is limited to 16 characters. The filename may be extended to a continuation line. All filenames should be entered in lower case. Nastran will convert to upper case, and the SOL 600 translator will reconvert to lower case. The creation runs should also use lower case for all external superelement punch filenames for case-sensitive computer systems.
9. If there are no elements in the residual (that is, all elements are in the external superelements, PARAM,MARCND99,-1 is required to output the displacements in the Marc .out file regardless of the specified Case Control request.
10. The ASMBULK option in the creation runs is required for SOL 600 when outr options are specified. It is recommended whether or not outr options are specified. All .asm files (created by the ASMBULK option) for all external superelements should be included in the Bulk Data before any punch files (produced using the DMIGPCH option). See the following input file examples.
11. At present, an OP2 with results datablocks only can be produced by a SOL 600 External Superelement residual execution. OP2 files which combine geometry and results datablocks cannot be produced. Other outr options such as xdb, f06 and punch are also not available for SOL 600 External Superelement residual runs.
12. For the External Superelement Initial run(s) to generate punch and asm files, no Case Control output requests should be made and the following DMAP should be inserted in the Executive Control in order to prevent extra information (which will cause an error) for being inserted into the punch file:

```
compile extout
alter 'sdr2' $
delete /iug1,,,,, $
alter 'sdr2' (2) $
delete /igulo,,,,, $
```

13. If data exists after ENDDATA, including any characters on the ENDDATA line, after the word ENDDATA in the creation run input, this data must be removed.

Typical File Setup for External Superelement Creation Run for SOL 600 (same as for other solution sequences)

```
SOL 101
compile extout
alter 'sdr2' $
delete /iug1,,,,, $
alter 'sdr2' (2) $
delete /iuglo,,,,, $
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO.
EXTSE2A
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
EXTSEOUT(ASMBULK,DMIGPCH,EXTID=100)
SPC = 100
BEGIN BULK
aset1,123456,840,thru,848
CORD2R,1001,1002,,,,,,1.0
(rest of file same as any other Nastran run)
```

### Typical File Setup for External Superelement Residual Run for SOL 600

```
SOL 600,101 outr=op2
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO. EXTSE2R
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
param,mextsee,1
SPC = 100
LOAD = 1000
DISP = ALL
K2GG=KAAX
M2GG=MAAX
BEGIN BULK
param,marcnd99,-1
force, 1000, 844, , 0.1, 0., 0., 1.
SPC1    100      12346     840      848
$2345678 2345678 2345678
mesuper      100 extse2a.pch
mesuper      200 extse2b.pch
include 'OUTDIR:extse2a.asm'
include 'OUTDIR:extse2b.asm'
include 'OUTDIR:extse2a.pch'
include 'OUTDIR:extse2b.pch'
ENDDATA
```

**METADATA**

Metadata for the input section

Defines the entry metadata.

**Format:**

| 1        | 2         | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|----------|-----------|------|---|---|---|---|---|---|----|
| METADATA |           | META |   |   |   |   |   |   |    |
|          | METAINFOR |      |   |   |   |   |   |   |    |

**Example:**

|          |             |                               |  |  |  |  |  |  |  |
|----------|-------------|-------------------------------|--|--|--|--|--|--|--|
| METADATA | Rear_Fender |                               |  |  |  |  |  |  |  |
|          |             | This is line 1 of metadata    |  |  |  |  |  |  |  |
|          |             | This is line 2 of metadata    |  |  |  |  |  |  |  |
|          |             | This is line 3 of metadata... |  |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| META       | Character handle of up to 24 characters identifying the metadata.                   |
| METAINFO   | A open-ended list of strings of 64 characters that represents the user information. |

**Remarks:**

1. META can include any character from defined bulk character set
2. METAINFO can contain basic Nastran characters and special symbols.

**MFLUID****Fluid Volume Properties**

Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

Format:

| 1      | 2    | 3       | 4   | 5   | 6      | 7      | 8      | 9      | 10 |
|--------|------|---------|-----|-----|--------|--------|--------|--------|----|
| MFLUID | SID  | CID     | ZFS | RHO | ELIST1 | ELIST2 | PLANE1 | PLANE2 |    |
|        | RMAX | FMEXACT |     |     |        |        |        |        |    |

Example:

|        |   |      |       |       |   |   |   |   |  |
|--------|---|------|-------|-------|---|---|---|---|--|
| MFLUID | 3 | 2    | 15.73 | 1006. | 3 | 4 | S | N |  |
|        |   | 100. |       |       |   |   |   |   |  |

| Descriptor        | Meaning  |
|-------------------|--|
| SID               | Set identification number. (Integer > 0)   |
| CID               | Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to $X_3$ ) and of planes of symmetry, if any. (Integer $\geq 0$ or blank)   |
| ZFS               | Intercept of the free surface on the $X_3$ axis of the coordinate system referenced by CID. If $X_3$ of a grid > ZFS then there is no fluid. See Remark 3. (Real; Default means that the free surface is located at an infinitely large positive value of XFS.)  |
| RHO               | Density of the fluid (Real).   |
| ELIST1            | Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on one side by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. See Remarks 3. and 5. (Integer $\geq 0$ )  |
| ELIST2            | Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on both sides by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. (Integer $\geq 0$ ; ELIST1 + ELIST2 $> 0$ )  |
| PLANE1,<br>PLANE2 | Planes of symmetry, antisymmetry, or no symmetry. "S" means that plane 1, which is the plane containing the $X_1$ and $X_3$ axes of CID, is a plane of symmetry. "A" means that plane 1 is a plane of antisymmetry. "N" means that it is neither. See Remark 5. Plane 2 uses "S", "A", or "N" for the $X_2$ and $X_3$ plane. (Character: "S", "A", or "N") |
| RMAX              | Characteristic length. Interactions between elements with separation that is greater than RMAX will be neglected. (Real $> 0.0$ ; Default = 1.0E 10)   |
| FMEXACT           | Exact integration is used if the distance between two elements is less than FMEXACT times the square root of the area of the larger element. Otherwise, center point integration is used by default. (Real; Default = 1.0E 15)   |

**Remarks:**

1. The MFLUID entry must be selected with the Case Control command MFLUID = SID.
2. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.
3. The wetted side of an element in ELIST1 is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal, as determined by applying the right-hand rule to the sequence of its corner points. The same element can appear on two ELIST entries, indicating that it forms a barrier between the unconnected fluids.
4. The fluid volume may be finite (interior) or infinite (exterior). The volume may be bounded by a free surface and one or two planes of structural symmetry. If structural symmetry is used, the structure must have the symmetric or antisymmetric boundary corresponding to the selection in fields 8 and 9. Interior fluids must have ELIST1 data and a free surface or plane of antisymmetry.
5. The planes of symmetry and/or antisymmetry defined in fields 8 and 9 must be planes of symmetry for the entire analysis. The user may apply appropriate structural boundary conditions at all grid points lying in these planes.
6. The current list of elements that may be placed in ELIST1 and ELIST2 include CTRIA3, CQUAD4, CTRIAR, and CQUADR.
7. The continuation entry is optional.
8. If there is ELIST1 data and no free surface nor plane of antisymmetry, the program assumes a special form of external fluid. These special external fluids must have a CID (field 3) such that the origin of the fluid coordinate system is near the center of the enclosed volume, since the singularity for volume change will be placed at the origin. Special external fluids are supported only in SOLs 103 and 107 through 112. If used in conventional solution sequences, System Fatal Message 3001 results for file 205.
9. See PARAM,VMOPT in “[Parameters](#)” on page 1409. VMOPT controls when the virtual mass is included in the mass matrix.
10. If any MFLUID entry is changed or added on restart then a complete re-analysis may be performed. Therefore, MFLUID entry changes or additions are not recommended on restart.
11. A tolerance is computed for each wetted element, with the value of  $0.01\sqrt{2A}$ , where A is the area of the element. If any grid point connected to the element lies within TOL below the free surface it is moved to the free surface.
12. Any element that has all grids on or above the free surface, after the grid points are moved by the procedures given in Remark 11. is removed from the ELIST. It is not included in the VM effects, and will produce no pressure output.

**MGRSPR**

Defines Grids to Add Soft Spring to Ground - SOL 600

This entry is used to add soft springs to ground at selected grids to stabilize the structure in a nonlinear analysis. It is most often used with contact to stabilize free-free bodies before they come into contact. The spring rates selected should be stiff enough to allow matrix decomposition but weak enough so they are not significant once full contact is achieved. Values on the order of 1.0E-6 to 1.0E-4 times the average main diagonal terms in the stiffness matrix for the grids selected are recommended.

**Format:**

| 1      | 2   | 3    | 4   | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|-----|------|-----|------|---|---|---|---|----|
| MGRSPR | ID1 | THRU | ID2 | IDIR | K |   |   |   |    |

**Example:**

|        |     |      |     |        |      |  |  |  |  |
|--------|-----|------|-----|--------|------|--|--|--|--|
| MGRSPR | 100 | THRU | 200 | 123    | 10.0 |  |  |  |  |
|        | 500 | THRU | 520 | 123456 | 1.0  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID1        | Starting grid ID. (Integer; Required; no Default)   |
| THRU       | Enter the character string THRU if more than one grid is desired.   |
| ID2        | Ending grid ID of the range. (Integer; or blank. If blank; ID2=ID1)   |
| IDIR       | Directions that the spring(s) will be added in, any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required) |
| K          | Spring rate. (Real; Required)   |

**Remarks:**

1. If springs are to be added to all grids, PARAM,MRSpring may be used instead of this entry.
2. Do not use PARAM,MRSpring and this entry in the same model.
3. Grids that do not exist in the range ID1 to ID2 will automatically not have springs in the Marc input.

**MINSTAT**

## Option to Define Initial State Variables for SOL 600

This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file. (MCHSTAT is used to define the temperatures that cause thermal strains). This entry may also be used to initialize other state variables if required.

**Format:**

| 1       | 2       | 3    | 4    | 5    | 6    | 7     | 8    | 9    | 10 |
|---------|---------|------|------|------|------|-------|------|------|----|
| MINSTAT | IDV     | IOPT |      | INCR |      | IFORM | IPRT | NPST |    |
|         | “FILE”  | Name |      |      |      |       |      |      |    |
|         | “ELEM”  | ELE1 | ELE2 | INT1 | INT2 | LAY1  | LAY2 | VAL  |    |
|         | “STATE” | NS   | IS1  | IS2  | IS3  | IS4   | IS5  | IS6  |    |
|         |         | IS7  | IS8  | IS9  | etc. |       |      |      |    |

**Example:**

|         |      |   |                       |   |  |   |  |   |  |
|---------|------|---|-----------------------|---|--|---|--|---|--|
| MINSTAT | 1    | 3 |                       | 1 |  | 0 |  | 9 |  |
|         | FILE |   | initial_state_example |   |  |   |  |   |  |

| Descriptor | Meaning   |
|------------|---|
| IDV        | State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.  |
| IOPT       | Option of how to enter the data. (Integer; Default = 3) (2,2) <ul style="list-style-type: none"> <li>1 Use the “ELEM” continuation line for as many elements as necessary</li> <li>2 Enter the data using user subroutine INITSV</li> <li>3 Read the data from a t16 or t19 file (see IFORM)</li> </ul> |
| INCR       | Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)  |
| IFORM      | Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) <ul style="list-style-type: none"> <li>0 Use binary (t16) file</li> <li>1 Use formatted (t19) file</li> </ul>  |
| IPRT       | Enter a value of 1 to suppress printing of state variable values defined in user subroutine INITSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)  |
| NPST       | Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9)<br>See MARCOUT for a list of the post codes.   |
| FILE       | Enter the character string FILE if IOPT=3. (Character; no Default; Required if IOPT=3)  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Enter the file name without the extension (.marc.t16 or .marc.t19). (Character; no Default)<br>The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file. |
| ELEM       | Enter the character string ELEM to start a list of elements and associated values if IOPT=1. (Character)  |
| ELE1       | First element with value VAL. (Integer; no Default; Required) (3,1)   |
| ELE2       | Last element with value VAL. (Integer; Default = ELE1) (3,2)  |
| INT1       | First integration point with value VAL. (Integer; no Default; Required) (3,3)   |
| INT2       | Last integration point with value VAL. (Integer; Default = INT1) (3,4)  |
| LAY1       | First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)   |
| LAY2       | Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)  |
| VAL        | New state value for these elements. (Real; no Default; Required) (4,1)  |
| STATE      | Enter the character string STATE to start a list of state variables. (Character)  |
| NS         | Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)   |
| ISi        | State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.   |

**Remarks:**

1. Only one MINSTAT entry may be entered in the input. If additional entries are found, the first will be used.
2. This entry maps to Marc's INITIAL STATE entry
3. (i,j) refer to Marc's INITIAL STATE (data block, field)
4. MINSTAT (and/or MCHSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a LOAD Case Control command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a LOAD Case Control command to reference it.

**MISLAND****SOL 600**

Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value in SOL 600 only.

**Format:**

| 1       | 2     | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-------|---|---|---|---|---|---|---|----|
| MISLAND | NELEM |   |   |   |   |   |   |   |    |

**Example:**

|         |    |  |  |  |  |  |  |  |  |
|---------|----|--|--|--|--|--|--|--|--|
| MISLAND | 20 |  |  |  |  |  |  |  |  |
|---------|----|--|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NELEM      | Number of elements to use as a limit for element deactivation. If the number of elements in an island is less than or equal to this number, all elements in this island are deactivated. (Integer; no Default; Required field) |

**Remarks:**

1. If this entry is made in the model, it subdivides the entire mesh into islands of connected regions. If the number of elements in a particular island is less than or equal to the specified value, all elements of this island are deactivated. This check is performed after element deactivation has taken place where the deactivation can be due to model input options, deactivation user subroutines, or through material damage or failure. Two elements are considered connected if they share a node for line elements, an edge for 2-D elements, or a face for 3-D solid elements. This option is useful for cases where unconnected elements or regions of elements might exist after the neighboring elements have been deactivated. There is no check performed to see if the island to be deactivated has enough boundary conditions. Only the number of elements in the island is used for determining if the elements should be deactivated.
2. Only one MISLAND entry should be placed in the input. If more than one is entered, the first will be used.

**MIXTURE****Constituents of "Composite" Material on Original and Potentially Damaged State - SOL 600**

Defines constituents of "composite" material on original and potentially damaged state. Used in SOL 600 only.

**Format:**

| 1       | 2   | 3      | 4     | 5      | 6  | 7      | 8    | 9      | 10 |
|---------|-----|--------|-------|--------|----|--------|------|--------|----|
| MIXTURE | MID | ITYPE  | IALPH | ISET   | M1 | FRACT1 | M2   | FRACT2 |    |
|         | M3  | FRACT3 | M4    | FRACT4 | M5 | FRACT5 | etc. |        |    |

**Example:**

|         |     |    |    |     |    |    |    |    |  |
|---------|-----|----|----|-----|----|----|----|----|--|
| MIXTURE | 120 | 3  | 1  | 200 | 10 | .2 | 20 | .2 |  |
|         | 30  | .2 | 40 | .2  | 50 | .2 |    |    |  |

| Descriptor | Meaning   |
|------------|---|
| MID        | Material ID (Integer; no Default, required field)   |
| ITYPE      | Mixture rule (Integer; Default = 1) <ul style="list-style-type: none"> <li>1 Weighted average of material properties based upon volume fraction.</li> <li>2 Weighted average of Hooke's law based upon volume fraction</li> <li>3 Weighted average of nonlinear stress strain curve based on volume fraction</li> </ul> |
| IALPH      | Flag controlling averaging procedure for thermal expansion (Integer; Default = 1) <ul style="list-style-type: none"> <li>1 Weighted average based upon volume fraction</li> <li>2 Not currently available</li> </ul>  |
| ISET       | ID of a SET3 containing the all the elements making up this material (Integer; no Default; Required)  |
| Mi         | Identification of a previously defined material, defined on a MATxxx entry (for example the MID on a MAT1 entry) (Integer; no Default)  |
| FRACTi     | Fraction for the ith component (see ITYPE). The sum of all FRACTi must add to 1.0 (Real; no Default)  |

**Remarks:**

1. Enter as many Mi, FRACTi components as desired. Each may have different properties, and failure criteria.
2. Fields 2-4 of each continuation entry are required, fields 5-9 may be left blank.

3. This option allows one to create a new material comprised of a number of other materials. The material behavior is based upon a "mixture" of the individual components using a mixture rule. Several of these mixture rules are only appropriate for linear elastic materials but also allow for temperature dependent material properties. The most sophisticated model (ITYPE=3) allows for the mixture of materials which undergo elastic-plastic behavior.
4. If void ratio or porosity is defined, it applied to all components in a uniform manner.
5. ITYPE 1 and 2 only support linear elastic material.
6. ITYPE 3 is not available using updated Lagrange - use PARAM,MARCPLAS,1 or PARAM,MARCPLAS,2 if any of the component materials have plasticity. The mixture will be limited to small strains, but large displacement, large rotation and follower forces and follower pressures can be included.
7. ITYPE 3 may not include the following material laws in any of the components.
  - Thermo-pore
  - Gurson damage
  - Simplified damage models 9 and 10
  - Gasket material
  - Shape memory material
  - Soils
  - User defined generalized stress strain
  - ORNL
  - Rigid-Plastic
  - Grain size effects
  - Rubber material (mooney, ogden, foam, gent, arruda-boyce)
  - Cohesive
8. Rebar elements and shell elements with offsets or non-composites with different properties for bending vs membrane (which use the Marc PSHELL option) may not be used for mixtures.
9. Within a layer, if the components are orthotropic or anisotropic, the preferred directions must be aligned.
10. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**MKAERO1****Mach Number - Frequency Table**

Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

**Format:**

| 1       | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|----|----|----|----|----|----|----|----|
| MKAERO1 | m1 | m2 | m3 | m4 | m5 | m6 | m7 | m8 |    |
|         | k1 | k2 | k3 | k4 | k5 | k6 | k7 | k8 |    |

**Example:**

|         |    |    |     |  |  |  |  |  |  |
|---------|----|----|-----|--|--|--|--|--|--|
| MKAERO1 | .1 | .7 |     |  |  |  |  |  |  |
|         | .3 | .6 | 1.0 |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| mi         | List of from 1 to 8 Mach numbers. (Real $\geq 0.0$ )     |
| kj         | List of from 1 to 8 reduced frequencies. (Real $> 0.0$ ) |

**Remarks:**

1. Blank fields end the list, and thus cannot be used for 0.0.
2. All combinations of (mi, kj) will be used.
3. The continuation entry is required.
4. Multiple MKAERO1 entries are permitted.
5. For the lifting surface theories (Doublet-Lattice and Mach Box), the maximum value of kj should be less than one quarter of the number of boxes on a representative chord (i.e.,  $\text{MAX}(kj) < \bar{C}/4\Delta x$  where  $\bar{C}$  is the reference chord and  $\Delta x$  is a typical box chord length).
6. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
7. The very low nonzero value of kj required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

**MKAERO2****Mach Number - Frequency Table**

Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

**Format:**

| 1       | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|----|----|----|----|----|----|----|----|
| MKAERO2 | m1 | k1 | m2 | k2 | m3 | k3 | m4 | k4 |    |

**Example:**

|         |     |     |     |     |     |     |     |     |  |
|---------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| MKAERO2 | .10 | .30 | .10 | .60 | .70 | .30 | .70 | 1.0 |  |
|---------|-----|-----|-----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning                              |
|------------|--------------------------------------|
| mi         | Mach numbers. (Real $\geq 0.0$ )     |
| ki         | Reduced frequencies. (Real $> 0.0$ ) |

**Remarks:**

1. MKAERO2 will cause the aerodynamic matrices to be computed for the given sets of parameter pairs. Embedded blank pairs are skipped.
2. No continuation entries are allowed, but multiple MKAERO2 entries are permitted.
3. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
4. The very low nonzero value of ki required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

**MLAYOUT****SOL 600**

Selects layered composite shell output to be placed in Marc's t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. The MARCOUT entry with LAYCODE of 3 or 103 must be used in conjunction with this entry. Used in SOL 600 only.

**Format:**

| 1       | 2  | 3    | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|---------|----|------|----|----|----|---|---|---|----|
| MLAYOUT | L1 | THRU | L2 | BY | L3 |   |   |   |    |
|         | L4 | THRU | L5 | BY | L6 |   |   |   |    |

**Example:**

|         |    |      |    |    |    |  |  |  |  |
|---------|----|------|----|----|----|--|--|--|--|
| MLAYOUT | 2  | THRU | 5  |    |    |  |  |  |  |
|         | 11 | THRU | 51 | BY | 10 |  |  |  |  |
|         | 52 | THRU | 54 |    |    |  |  |  |  |
|         | 21 | THRU | 20 | BY | 10 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| L1, L4     | Starting layer of a sequence. (Integer; Required; no Default) |
| L2, L5     | Ending layer of a sequence. (Integer; Default = L1 or L4)     |
| L3, L6     | Increment value for the sequence. (Integer; Default = 1)      |

**Remarks:**

1. This entry must be used in conjunction with MARCOUT with LAYCODE=3 or 103.
2. L2 must be larger than L1.
3. L1, L4, etc. must be positive integers of 1 or larger.
4. L2, L5 should not exceed the largest number of layers in the model.
5. Selection of all layers can lead to extremely large output files.
6. In the example, L1 could also be 1 instead of 2 and the largest layer could be 55 instead of 54 with causing an error.
7. If the layers designated on this entry are to apply to selected elements, enter the elements on the T16SEL entry. It is not currently possible to process different layered output for multiple groups of elements (i.e., it is not currently possible to request output of layers 1-20 for some elements and 1-10 for others.)

**MNF600**

Defines Auxiliary Data for MSC Adams MNF Files - SOL 600

Generated by the Marc portion of a SOL 600 execution. Used in SOL 600 only. (See also the MDMIOUT Bulk Data entry.)

**Format:**

| 1      | 2  | 3       | 4       | 5      | 6     | 7     | 8     | 9     | 10 |
|--------|----|---------|---------|--------|-------|-------|-------|-------|----|
| MNF600 | ID | ISTRESS | ISTRAIN | ISHELL | MASSU | LENGU | TIMEU | FORCU |    |

**Example:**

|        |     |   |   |   |   |   |   |   |  |
|--------|-----|---|---|---|---|---|---|---|--|
| MNF600 | 100 | 1 | 1 | 3 | 2 | 7 | 3 | 2 |  |
|--------|-----|---|---|---|---|---|---|---|--|

| Descriptor     | Meaning   |
|----------------|---|
| ID             | Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. Must match the MDMIOUT entry. At present, ID is ignored and the first entry will apply to all subcases. (Integer; Default = 1) |
| ISTESS (2)     | Flag to compute stress and place in the MNF file (0=no stress, 1=stress) (Integer; Default = 0)   |
| ISTRAIN (3)    | Flag to compute strain and place in the MNF file (0=no strain, 1=strain) (Integer; Default = 0)   |
| ISHELL (4)     | For shell elements, this entry describes which location the stresses or strains will be output to the MNF file. 1=top, 2=center, 3=bottom (Integer; Default = 1)  |
| MASSU<br>[2,1] | Mass units for MNF file. (Integer; Default = 1: kilogram) The following possible values may be entered:<br>1:kilogram (Default)<br>2:pound mass<br>3:slug<br>4:gram<br>5:ounce mass<br>6:kpound mass<br>7:megagram<br>8:dozen slug  |

| Descriptor      | Meaning   |
|-----------------|---|
| LENGU           | Length units for MNF file (Integer; Default = 2: meter) The following possible values may be entered:<br>1:kilometer<br>2:meter (Default)<br>3:centimeter<br>4:millimeter<br>5:mile<br>6:foot<br>7:inch                     |
| TIMEU<br>[2,3]  | Time units for MNF file (Integer; Default = 3: second). The following possible values may be entered:<br>1:hour<br>2:minute<br>3:second (Default)<br>4:millisecond  |
| FORCEU<br>[2,4] | Force units for MNF file (Integer; Default = 1: newton). The following possible values may be entered:<br>1:Newton (Default)<br>2:poind force<br>3:kilogram force<br>4:ounce force<br>5:dyne<br>6:kNewton<br>7:kpound force |

**Remarks:**

1. The MDMIOUT entry is the primary entry which generates an MNF file. This entry, MNF600 is only necessary if one or more of the fields is required to define non-default values, for example to generate stresses or strains or to specify the units.
2. The ID must be the same as the MDMIOUT ID.
3. Only one MNFDAT entry is allowed in an input file. If more than one is entered, the first will be used.
4. All remarks concerning MNF files for the MDMIOUT entry are also applicable to this entry.
5. (i) Indicates the corresponding field of Marc's MNF Parameter.
6. [i,j] Indicates the corresponding datablock and field of Marc's MNF units entry.

**MODTRAK****Mode Tracking Parameters**

Specifies parameters for mode tracking in design optimization (SOL 200).

**Format:**

| 1       | 2   | 3      | 4       | 5        | 6 | 7 | 8 | 9 | 10 |
|---------|-----|--------|---------|----------|---|---|---|---|----|
| MODTRAK | SID | LOWRNG | HIGHRNG | MTFILTER |   |   |   |   |    |

**Example:**

|         |     |   |    |      |  |  |  |  |  |
|---------|-----|---|----|------|--|--|--|--|--|
| MODTRAK | 100 | 1 | 26 | 0.80 |  |  |  |  |  |
|---------|-----|---|----|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Sets identification number that is selected in the Case Control Section with the MODTRAK command. See Remark 1. (Integer; no Default)            |
| LOWRNG     | Lowest mode number in range to search. See Remark 2. (Integer $\geq 0$ ; Default = 0. If nonzero, LOWRNG < HIGHRNG.)                             |
| HIGHRNG    | Highest mode number in range to search. See Remark 2. (Integer $> 0$ ; Default = number of eigenvalues extracted. If nonzero, LOWRNG < HIGHRNG.) |
| MTFILTER   | Filtering parameter used in mode cross-orthogonality check. See Remark 3. (Real; Default = 0.9)  |

**Remarks:**

1. Only the designed modes for the subcase will be tracked. A designed mode is one that is used in the design model (in connection with either objective or constraints) and, therefore, identified on a DRESP1 entry.
2. The range of modes LOWRNG through HIGHRNG, inclusive, will be used to track the designed modes. If LOWRNG and HIGHRNG are both blank, then all computed modes will be used to search for the designed modes. Since large numbers of computed modes will result in higher computational costs, limiting the search range with LOWRNG and HIGHRNG is recommended.
3. Modes are considered to correlate if their mass normalized cross orthogonalities are greater than MTFILTER.

**MOMAX**

## Conical Shell Static Moment

Defines a static concentrated moment load on a ring of a conical shell.

Format:

| 1     | 2   | 3   | 4   | 5 | 6  | 7  | 8  | 9 | 10 |
|-------|-----|-----|-----|---|----|----|----|---|----|
| MOMAX | SID | RID | HID | S | MR | MP | MZ |   |    |

Example:

|       |   |   |   |     |     |     |     |  |  |
|-------|---|---|---|-----|-----|-----|-----|--|--|
| MOMAX | 1 | 2 | 3 | 1.0 | 0.1 | 0.2 | 0.3 |  |  |
|-------|---|---|---|-----|-----|-----|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| RID        | Ring identification number. See the RINGAX entry. (Integer > 0)   |
| HID        | Harmonic identification number or a sequence of harmonics. See Remark 5.<br>(Integer $\geq 0$ or Character) |
| S          | Scale factor. (Real)  |
| MR, MP, MZ | Moment components in the r, $\phi$ , z directions. (Real)   |

Remarks:

1. MOMAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD=SID.
3. A separate entry is needed for the definition of the moment associated with each harmonic.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2", where n1 is the start of the sequence and n2 is the end of the sequence; i.e., for harmonics 0 through 10, the field would contain "S0T10".

**MOMENT****Static Moment**

Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

Format:

| 1      | 2   | 3 | 4   | 5 | 6  | 7  | 8  | 9 | 10 |
|--------|-----|---|-----|---|----|----|----|---|----|
| MOMENT | SID | G | CID | M | N1 | N2 | N3 |   |    |

Example:

|        |   |   |   |     |     |     |     |  |  |
|--------|---|---|---|-----|-----|-----|-----|--|--|
| MOMENT | 2 | 5 | 6 | 2.9 | 0.0 | 1.0 | 0.0 |  |  |
|--------|---|---|---|-----|-----|-----|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| G          | Grid point identification number at which the moment is applied. (Integer > 0)   |
| CID        | Coordinate system identification number. (Integer $\geq 0$ or blank)   |
| M          | Scale factor. (Real)   |
| Ni         | Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq 0.0$ unless M is zero) |

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{m} = M \vec{N}$$

where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\vec{m}$  is equal to M times the magnitude of  $\vec{N}$ .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body: the MID of a rigid material (MATRIG) or the EID of a RBE2D. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2D and a rigid material are used with these TYPES. SOL 700 only.

**MOMENT1****Follower Moment, Alternate Form 1**

Defines a concentrated moment at a grid point by specifying a magnitude and two grid points that determine the direction.

Format:

| 1       | 2   | 3 | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|---------|-----|---|---|----|----|---|---|---|----|
| MOMENT1 | SID | G | M | G1 | G2 |   |   |   |    |

Example:

|         |   |    |       |    |    |  |  |  |  |
|---------|---|----|-------|----|----|--|--|--|--|
| MOMENT1 | 6 | 13 | -2.93 | 16 | 13 |  |  |  |  |
|---------|---|----|-------|----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| G          | Grid point identification number at which the moment is applied. (Integer > 0)  |
| M          | Magnitude of moment. (Real)   |
| G1, G2     | Grid point identification numbers used to define the unit vector $\vec{n}$ . (Integer > 0; G1 and G2 cannot be coincident.) |

Remarks:

1. The concentrated moment applied to grid point G is given by

$$\vec{m} = M \vec{n}$$

where  $\vec{n}$  is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

**MOMENT2**

## Follower Moment, Alternate Form 2

Defines a concentrated moment at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

| 1       | 2   | 3 | 4 | 5  | 6  | 7  | 8  | 9 | 10 |
|---------|-----|---|---|----|----|----|----|---|----|
| MOMENT2 | SID | G | M | G1 | G2 | G3 | G4 |   |    |

Example:

|         |   |    |       |    |    |    |    |  |  |
|---------|---|----|-------|----|----|----|----|--|--|
| MOMENT2 | 6 | 13 | -2.93 | 16 | 13 | 17 | 13 |  |  |
|---------|---|----|-------|----|----|----|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| G          | Grid point identification number at which the moment is applied. (Integer > 0)   |
| M          | Magnitude of moment. (Real)  |
| Gi         | Grid point identification numbers used to determine the unit vector $\vec{n}$ . (Integer > 0; G1 and G2 cannot be coincident; G3 and G4 cannot be coincident.) |

Remarks:

1. The concentrated moment applied to grid point G is given by

$$\vec{m} = M \vec{n}$$

where  $\vec{n}$  is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

**MONCARL****Parameters for Monte-Carlo simulation**

Parameters for conducting Monte-Carlo simulation using the non-parametric variability method (NPVM).

**Format:**

| 1       | 2      | 3      | 4     | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|--------|--------|-------|-------|---|---|---|---|----|
| MONCARL | SID    | VARTYP |       |       |   |   |   |   |    |
|         | STRVAR | MSVAR  | KSVAR | BSVAR |   |   |   |   |    |
|         | FLUVAR | MFVAR  | KFVAR | BFVAR |   |   |   |   |    |

**Example:**

|         |        |          |       |       |  |  |  |  |  |
|---------|--------|----------|-------|-------|--|--|--|--|--|
| MONCARL | 100    | DELTAVAR |       |       |  |  |  |  |  |
|         | STRVAR | 0.001    | 0.05  | 0.002 |  |  |  |  |  |
|         | FLUVAR | 0.0      | 0.001 | 0.0   |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number referenced by MONCARL case control. (Integer > 0; Required)  |
| VARTYP     | Select the type of variabilities that follow (Character; Default = ALPHAVAR)<br>ALPHAVER Variabilities in this entry are Alpha variabilities.<br>DELTAVAR Variabilities in this entry are Delta variabilities. |
| “STRVAR”   | Indicates Structural variability parameters follow. (String)   |
| MSVAR      | Structural mass variability. (Real >= 0.0; Default = 0.0)  |
| KSVAR      | Structural stiffness variability. (Real >= 0.0; Default = 0.0)   |
| BSVAR      | Structural damping variability. (Real >= 0.0; Default = 0.0)   |
| “FLUVAR”   | Indicates Fluid variability parameters follow (String)   |
| MFVAR      | Fluid mass variability. (Real >= 0.0; Default = 0.0)   |
| KFVAR      | Fluid stiffness variability. (Real >= 0.0; Default = 0.0)  |
| BFVAR      | Fluid damping variability. (Real >= 0.0; Default = 0.0)  |

### Remarks

1. Please use below as the guideline for selecting above mentioned structural and fluid alpha variabilities (which should be > 0.0):
  - a. Low variability level = 0.001
  - b. Medium variability level = 0.01

- c. High variability level = 0.05
- 2. Alpha variabilities are related to delta variabilities through the order of the matrix to be randomized,  $n$  (or equivalently the number of modes),

$$a = \delta \sqrt{\frac{2}{n+1}}$$

- 3. There is a limitation in the admissible values for the variability level. The maximum value for delta variability equals,

$$\delta_{\max} = \sqrt{\frac{n+1}{n+5}}$$

On a model with a large amount of modes ( $n > 1$ ), the maximum value of delta tends to 1 ( $\delta_{\max} \approx 1$ ).

The maximum value of alpha variability using the above equations equals,

$$a_{\max} = \sqrt{\frac{2}{n+5}}$$

- 4. An unphysical (larger than permissible or negative value) selection of a variability would results in a FATAL.

**MONCNCM****Nondimensional Normal Force and Pitching Moment**

Provides a stripwise aerodynamic normal force and pitching moment coefficients for CAERO1 based aerodynamics.

**Format:**

| 1       | 2    | 3       | 4       | 5   | 6       | 7 | 8 | 9 | 10 |
|---------|------|---------|---------|-----|---------|---|---|---|----|
| MONCNCM | NAME | LABEL   |         |     |         |   |   |   |    |
|         | MREF | CAERID1 | CAERID2 | ... | CAERIDn |   |   |   |    |

**Example:**

|         |      |  |      |  |  |  |  |  |  |
|---------|------|--|------|--|--|--|--|--|--|
| MONCNCM | LEFT | Normal force and pitching moment values for strips on the left wing. |      |  |  |  |  |  |  |
|         |      | 2001   | 3001 |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Unique character string of up to 8 characters identifying the family of chord-wise strips. (Character)  |
| LABEL      | A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the component. (Optional)                                    |
| MREF       | Fractional chord location of the aerodynamic strip about which the pitching moment is to be calculated. (Real; $\leq 0.0$ MREF $\leq 1.0$ ; Default = 0.25) |
| CAERID1    | ID of a CAERO1 entry that contains aero boxes for which strip results are to be produced. (Integer > 0 or “ALL”). See Remarks 2. and 3.                     |

**Remarks:**

1. The LABEL is optional.
2. Output is produced for all chordwise strips on the referenced CAERO1 entries. If a strip spans CAERO1 panels, results are output for the total strip.
3. If CAERID1 is the character string “ALL”, then output will be produced for all strips.
4. Strips are identified based on the y and z coordinates in the aerodynamic coordinate system.
5. If strips from separate CAERO1’s have the same y and z coordinate and the same strip width and share an xlocation (i.e., the leading edge of one strip equals the trailing edge from another) then they are processed as a single strip.
6. The normal force component is normalized by the dynamic pressure times the surface area of the strip. The moment component is normalized by the dynamic pressure times the surface area of the strip times the chord length at the center of the strip. The moment is calculated about the MREF location of the strip with the quarter-chord of the strip as the default.
7. Only CAERO1 id’s are supported.

**MONDSP1****Displacement Monitor Point**

Defines a virtual point displacement response at a user-defined reference location (coordinates and coordinates system) as a weighted average of the motions at a set of grid points.

**Format:**

| 1       | 2    | 3     | 4  | 5 | 6 | 7 | 8  | 9      | 10 |
|---------|------|-------|----|---|---|---|----|--------|----|
| MONDSP1 | NAME | LABEL |    |   |   |   |    |        |    |
|         | AXES | COMP  | CP | X | Y | Z | CD | INDDOF |    |

**Example:**

|         |         |                            |      |     |       |      |      |  |  |
|---------|---------|----------------------------|------|-----|-------|------|------|--|--|
| MONDSP1 | WING195 | Wing twist at station 150. |      |     |       |      |      |  |  |
|         | 5       | WING150                    | 1001 | 120 | 150.0 | 17.0 | 1002 |  |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Character string of up to 8 characters identifying the monitor point (Character)  |
| LABEL      | A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.   |
| AXES       | Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)  |
| COMP       | The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined.   |
| CP         | The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer $\geq 0$ ; Default = 0)   |
| X,Y,Z      | The coordinates in the CP coordinate system at which the displacement is to be monitored.   |
| CD         | The identification number of a coordinate system in which the resulting displacement components are output. (Integer $\geq 0$ ; Default = the coordinate system specified by the CP field)                                      |
| INDDOF     | Component numbers of all the independent grids from which the derived, dependent, monitor DOF's are to be computed. (Any unique combination of the integers 1 through 6 with no embedded blanks.) See Remark 3. (Default = 123) |

**Remarks:**

1. The MONDSP1 is available for SOLs 101, 103, 105, 108, 109, 111, 112, 144, 146 and 200.
2. The entry can create either an aerodynamic or a structural MONDSP1, depending on the data provided in the COMP field. A structural MONDSP1 can have the same name as an aerodynamic MONDSP1, but two structural or two aerodynamic MONDSP1s cannot have the same name.

3. The INDDOF field defines the Ci field on the virtual RBE3; that is, it defines the components of the grids on the AECOMP that will be sampled to define the dependent (monitor point) displacement. Typically, the default is the correct choice. However, if there is only a single grid point, all six DOF's can be used.

**MONGRP****Specification of a Monitor Group**

Defines a collection of monitor points into a group available for postprocessing.

**Format:**

| 1      | 2       | 3        | 4       | 5      | 6         | 7 | 8 | 9 | 10 |
|--------|---------|----------|---------|--------|-----------|---|---|---|----|
| MONGRP | GNAME   | LABEL    |         |        |           |   |   |   |    |
|        | NAME1   | CLASS1   | REAL1   | INT1   | STRING1   |   |   |   |    |
|        | NAME2   | CLASS2   | REAL2   | INT2   | STRING2   |   |   |   |    |
|        | NAME(m) | CLASS(m) | REAL(m) | INT(m) | STRING(m) |   |   |   |    |

**Example:**

|        |         |  |      |   |        |  |  |  |
|--------|---------|--|------|---|--------|--|--|--|
| MONGRP | WING    | A COLLECTION OF MONITOR POINTS ON THE WING |      |   |        |  |  |  |
|        | WING195 | SMONPT1                                    |      | 1 | WL195  |  |  |  |
|        | WING205 | SMONPT1                                    |      | 2 | WL205  |  |  |  |
|        | WINGDSP | SMONDPI                                    | 20.0 |   | GRID20 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| GNAME      | A character string of up to 8 characters identifying the monitor group. (Character)                               |
| LABEL      | A String comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point. |
| NAME(i)    | Name of an existing monitor point. (Character)  |
| CLASS(i)   | Type of monitor point (either AMONPT1, AMONDPI, SMONPT1, SMONDPI, MONPT2, MONPT3)                                 |
| REAL(i)    | Optional real value to designate a real property of the monitor point.  |
| INT(i)     | Optional integer value to designate a integer property of the monitor point.                                      |
| STRING     | A label of the current monitor type. (Character string of no more than 32 character)                              |

**Remarks:**

1. The MONGRP entry provides the user a means of specifying groups of monitor point data but does not produce any output on its own.
2. Aerodynamic (AMONPT1 and AMONDPI) and structural (SMONPT1, SMONDPI, MONPT2, MONPT3) cannot be mixed in a given group.
3. The REAL(i), INT(i) and STRING(i) data are provided to facilitate user display of the monitor data and can be used, for example, to label xy plots.
4. The LABEL should be unique across MONGRP entries.

**MONPNT1****Integrated Load Monitor Point**

Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system. The integrated loads about this point over the associated nodes will be computed and printed for statics, dynamics and static aeroelastic trim analyses and form integrated loads on the nonlinear static aeroelastic database.

**Format:**

| 1       | 2    | 3     | 4  | 5 | 6 | 7 | 8  | 9 | 10 |
|---------|------|-------|----|---|---|---|----|---|----|
| MONPNT1 | NAME | LABEL |    |   |   |   |    |   |    |
|         | AXES | COMP  | CP | X | Y | Z | CD |   |    |

**Example:**

|         |         |                                     |  |     |       |      |  |  |  |
|---------|---------|-------------------------------------|--|-----|-------|------|--|--|--|
| MONPNT1 | WING155 | Wing Integrated Load to Butline 155 |  |     |       |      |  |  |  |
|         | 34      | WING                                |  | 0.0 | 155.0 | 15.0 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| NAME       | Character string of up to 8 characters identifying the monitor point (Character; no Default)   |
| LABEL      | A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point. (Character; optional)  |
| AXES       | Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)   |
| COMP       | The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined. (Character; no Default)  |
| CP         | The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer $\geq 0$ ; Default = 0)  |
| X,Y,Z      | The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).  |
| CD         | The identification number of a coordinate system in which the resulting load components are output. (Integer $\geq 0$ ; Default = the coordinate system specified by the CP field) |

**Remarks:**

1. The MONPNT1 is available for SOLs 101, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The entry can create either an aerodynamic or a structural MONPNT1, depending on the data provided in the COMP field. A structural MONPNT1 can have the same name as an aerodynamic MONPNT1, but two structural or two aerodynamic MONPNT1s cannot have the same name.

3. In SOL 144, static aeroelasticity, the structural MONPNT1 integration is modified to account for the redistribution of loads and inertia caused by MPCs and rigid elements. In all other supported solution sequences, the integration is done using the g-set loads on all grid points specified on the referenced SET1 entry.

**MONPNT2****Internal Load Monitor Point**

Element Monitor Output Results Item.

**Format:**

| 1       | 2     | 3    | 4        | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|-------|------|----------|-------|---|---|---|---|----|
| MONPNT2 | NAME  |      |          | LABEL |   |   |   |   |    |
|         | TABLE | TYPE | NDDLitem | EID   |   |   |   |   |    |

**Example:**

|         |        |                               |      |     |  |  |  |  |
|---------|--------|-------------------------------|------|-----|--|--|--|--|
| MONPNT2 | SB100  | Leading edge stringer at root |      |     |  |  |  |  |
|         | STRESS | CBAR                          | SX2A | 100 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| NAME       | Unique character string of up to 8 characters identifying the monitor point (Character; no Default)  |
| LABEL      | An optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point. (Character; Default = Blank) |
| TABLE      | Type of output to be monitored. Options are STRESS, FORCE or STRAIN. (Character; no Default)   |
| TYPE       | Element type (Character; no Default)   |
| NDDLitem   | Component for this type to be monitored. This is the NDDL label for the particular Table and element type. (Character; no Default)               |
| EID        | Element ID. (Integer > 0)  |

**Remarks:**

1. The MONPNT2 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146 and 200.
2. Most element types have some items that can be monitored.
3. An assumption is made that the desired component is linear with respect to the displacement vector. If this assumption is not valid, the results will be incorrect.
4. Fictitious Table/Type/NDDLItems/EID generate a warning message and are ignored.
5. NDDL descriptions for Table=FORCE can be found in the [MSC Nastran DMAP Programmer's Guide](#) within the OEF datablock description. Table=STRESS and STRAIN are contained in the OES datablock description.

Once within the datablock description you can search for the element name (or better yet, element number, see the following table) you are interested in. There can be several different descriptions for an element type. For example, real vs. complex, thermal, stress vs. strain (within the OES description), linear vs nonlinear. In addition, the shell class of elements (quad4, quad8, quadr, tria3, tria6, triar) will have a composite form (quad4lc, quud8lc, quadrlc, tria3lc, .., i.e., basename + "LC"), corner or bilin stresses (basename + "C").

By looking at the comments contained in the text make sure you are reading from the appropriate section. The NDDLItem is labeled as the 'NAME' field within the [MSC Nastran DMAP Programmer's Guide](#).

You can also print the NDDL description for the entire database by running the following 4 statement bulk data file.

```
sol loadnndl  
compile nndl=nndl,list  
cend  
enddata
```

**MONPNT3****Integrated Load Monitor Point**

Sums select Grid Point Forces to a user chosen monitor point.

**Format:**

| 1       | 2    | 3       | 4       | 5  | 6 | 7 | 8 | 9     | 10 |
|---------|------|---------|---------|----|---|---|---|-------|----|
| MONPNT3 | NAME | LABEL   |         |    |   |   |   |       |    |
|         | AXES | GRIDSET | ELEMSET | CP | X | Y | Z | XFLAG |    |
|         | CD   |         |         |    |   |   |   |       |    |

**Example:**

|         |        |                       |   |   |      |  |  |     |  |
|---------|--------|-----------------------|---|---|------|--|--|-----|--|
| MONPNT3 | t0     | Fuselage station 1105 |   |   |      |  |  |     |  |
|         | 123456 | 1                     | 2 | 0 | 30.0 |  |  | ASM |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Unique character string of up to 8 characters identifying the monitor point (Character; Required)   |
| LABEL      | A optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point.  |
| AXES       | Component axes about which to sum. Any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required)                              |
| GRIDSET    | Refers to a SET1 entry that has a list of grids to be included in the monitored point. (Integer; Required)  |
| ELEMSET    | Refers to a SET1 entry that has a list of elements to include at the monitored point. (Integer; optional)   |
| CP         | The identification number of a coordinate system in which the (x,y,z) coordinates are defined. (Integer $\geq 0$ ; Default=0)                                   |
| X,Y,Z      | The coordinates in the CP coordinate system about which the forces are to be summed. (Real; Default = 0.0).   |
| XFLAG      | Exclusion flag. Exclude the indicated Grid Point Force types from summation at the monitor point. Default = blank (no type excluded). See Remark 4.             |
| S          | SPCforces   |
| M          | MPC forces  |
| A, L, or P | applied loads   |
| D          | dmig's (and any other type not described above) at the monitored point.   |
| C          | contact forces (SOL 400 only).  |
| CD         | The identification number of a coordinate system in which the results are output. (Integer $>= 0$ , Default = the coordinate system specified by the CP field.) |

**Remarks:**

1. The MONPNT3 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN). Note that MONPNT3 output to csv file is available only to SOL 144.
  2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
  3. Fictitious grids or elements do not produce error or warning messages.
  4. For linear solution sequences, if the exclusion flags omit some grid point force types but not all of them (i.e; if the field is not blank and is not SMAD) then the following limitations exist:
    - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
    - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
    - It is only available in SOLs 101, 103, 144 and SOL 200 (ANALYSIS = STATICS, MODES, or SAERO).
  5. MONPNT3 can be useful in calculating shear, moment and torque from the internal loads in a structure. For example, if a split is made in a fuselage component and all the grids that reside on this split are placed in the GRIDSET, differing internal loads resultants can be obtained based on the ELEMSET and XFLAG values. If a cut is made in a structure, there are three types of loads:
    - A - loads that come from the elements upstream of the cut.
    - B - loads that are applied to the grids on the cut from any other source.
    - C - loads that come from the elements downstream of the cut.Where A + B + C = 0
- Useful options for these two values are:
- a. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is blank then the internal load is calculated using the elements that connect to the GRIDSET that are NOT included in ELEMSET. The direction of this load is reversed. So actually -C will be calculated which matches A+B. This in effect gives the resultants on the downstream/inboard side of the split pointing into upstream/ outboard direction which will include any loads applied to the GRIDSET from any source.
  - b. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is SMAD then the internal load is calculated using the elements listed in ELEMSET. In that case, A will be calculated. This in effect gives the resultants on the upstream/outboard side of the split and does not include loads applied to the GRIDSET from any other source.
6. For SOL 400 (ANALYSIS = NLSTAT or NLTRAN), the following limitations exist:
    - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
    - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
    - Thermal loads are not supported.

7. For linear solution sequences with XFLAG= blank or SMAD, the program applies a mini-EMA (element matrix assembly) algorithm that bypasses GPFDR (grid point force data recovery) and another modules to improve performance. This algorithm forms a stiffness matrix that just includes those elements specified in MONPNT3 entries. The monitor point results are computed directly from multiplying this matrix by an integration matrix that transfers forces to the monitor points. These results may be slightly different from the GPFDR results, such as clean zero forces versus small residual forces.

**MONSUM**

## Linear Combinations of Monitor Point Components

Defines a new monitor result that is the weighted sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (see Remark 5.).

Format:

| 1      | 2        | 3        | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|----------|----------|--------|--------|--------|--------|--------|--------|----|
| MONSUM | NAME     | LABEL    |        |        |        |        |        |        |    |
|        | NEWAXISA | MTYPE1A  | NAME1A | AXES1A | COEF1A | NAME2A | AXES2A | COEF2A |    |
|        |          |          | NAME3A | AXES3A | COEF3A | etc.   |        |        |    |
|        |          | MTYPE2A  | NAME2A | etc.   |        |        |        |        |    |
|        | NEWAXISB | MTYPE1 B | NAME1B | AXES1B | COEF1B | NAME2B | etc.   |        |    |
|        |          | MTYPE1C  | etc.   |        |        |        |        |        |    |

Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

|        |         |  |       |   |       |       |   |       |  |
|--------|---------|--|-------|---|-------|-------|---|-------|--|
| MONSUM | AM1PSM3 | Adding the wing lift to a free body load |       |   |       |       |   |       |  |
|        | 3       | AMONPNT1                                 | WING  | 3 | 1.0   |       |   |       |  |
|        |         | MONPNT3                                  | FREEW | 3 | 1400. | FREEW | 5 | -1000 |  |

Alternate Format:

| 1      | 2     | 3        | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|-------|----------|--------|--------|--------|--------|--------|--------|----|
| MONSUM | NAME  |          |        |        | LABEL  |        |        |        |    |
|        | MTYPE | NEWAXISA | NAME1A | AXES1A | COEF1A | NAME2A | AXES2A | COEF2A |    |
|        |       |          | NAME3A | AXES3A | COEF3A | etc.   |        |        |    |
|        |       | NEWAXISB | NAME2A | AXES1B | COEF1B | NAME2B | AXIS2B | COEF2B |    |
|        |       |          | NAME3B | AXES3B | COEF3B | etc.   |        |        |    |
|        |       | NEWAXISC | etc.   |        |        |        |        |        |    |

Alternate Example: Scale an existing monitor point

|        |          |  |      |     |       |      |     |        |  |
|--------|----------|--|------|-----|-------|------|-----|--------|--|
| MONSUM | ROOT     | Scale from in-lbs to newton-cm and from lbs to newtons |      |     |       |      |     |        |  |
|        | AMONPNT1 | 123456   | ROOT | 123 | 4.482 | ROOT | 456 | 11.385 |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Character string of up to 8 characters identifying the monitor result. (Character)  |
| LABEL      | A string comprising of no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.  |
| NEWAXISj   | The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 7. and 8.) |
| MTYPij     | Monitor type to be merged. (Character, one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6.  |

| Descriptor | Meaning  |
|------------|--|
| MTYPE      | Monitor type to be merged in the alternate format. (Character; one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6. |
| NAMEij     | Name of the monitored quantity that is to be merged  |
| AXESij     | Component axes that are to be summed. (any unique combination of the integers 1 to 6 with no embedded blanks)  |
| COEFij     | Coefficient to be applied to the component(s) called out on AXESij field. (Real; Default = 1.0)  |

**Remarks:**

1. The MONSUM is available in SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The LABEL is a 56 character string that should be unique.
3. The MONSUM can be used to update an existing monitor result by setting all the NAMEij terms equal to NAME. In this case, the alternate format is available and the NEWAXISj component is a scalar multiple of the original component:  
$$\text{MONSUM}_j = M_{Rj} \text{ COEF}_j$$
4. When the NAME differs from the NAMEij values, it must be unique with respect to all monitor quantities. The result of the MONSUM entry is to create new monitor point(s) that are equal to:

$$\text{MONSUM}_j = \sum_i^n \text{COEF}_{ij} \text{MR}_{ij}$$

where  $\text{MR}_{ij}$  is the result from the individual component.

5. The merged monitor points must be of a similar type. “Similar” types are defined as:  
Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3  
Average displacement monitor points: AMONDSP1 and SMONDSP1
6. For MONPNT1’s and MONDSP1’s, the MTYPij can be aerodynamic or structural.  
MTYPij=AMxxx1 designates aerodynamic while SMxxx1 designates a structural monitor point.
7. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are determined from NEWAXISj.
8. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6.
9. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM entry.
10. Structural monitor points may span superelements.

11. If multiple types are specified on a MONSUM, the resulting entry is of a type that appears on the MONSUM with the following order of precedence: smonpnt1, smondsp1, monpnt3, amonpnt1, amondsp1. E.g., a amonpnt1 and a monpnt3 appearing on the same entry will result in a monpnt3 regardless of which appears first.
12. If all the MTYPij values are of the same type, the alternate format provides a simplified interface.
13. A MONSUM can reference another MONSUM (including itself) as long as there is not a circular reference. A MONSUM cannot reference another MONSUM1.
14. MONPNT3's that have one, two or three excluded items (e.g., XFLAG=SMA) cannot be referenced on a MONSUM.
15. Since CP,X,Y,Z and CD are not known with certainty for a MONSUM, they are not printed as part of the monitor point output. Further, in SOL's 101 and 144, COMP=\*\*SUM\*\*, is used to indicate that the associated monitor point has been derived from a MONSUM.

**MONSUM1**

Linear combination of two or more monitor points.

Defines a new monitor result that is the weighted sum of existing monitor results. The location of the computed MONSUM1 is specified. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3.)

**Format:**

| 1       | 2       | 3     | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|---------|-------|--------|--------|--------|--------|--------|--------|----|
| MONSUM1 | NAME    | LABEL |        |        |        |        |        |        |    |
|         | CP      | X     | Y      | Z      | CD     |        |        |        |    |
|         | NEWAXES | MTYP1 | NAME1a | AXES1A | COEF1A | NAME2A | AXES2A | COEF2A |    |
|         |         |       | NAME3A | AXES3A | COEF3A | Etc    |        |        |    |
|         |         | MTYP2 | NAME2A | AXES1B | COEF1B | NAME2B | AXIS2B | COEF2B |    |
|         |         |       | NAME3B | AXES3B | COEF3B | Etc.   | .      |        |    |
|         |         | MTYP3 | Etc    |        |        |        |        |        |    |

**Example:** Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

|         |        |  |         |      |        |         |     |         |  |
|---------|--------|--|---------|------|--------|---------|-----|---------|--|
| MONSUM1 | ROOT   | SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS |         |      |        |         |     |         |  |
|         |        | 500.   | 0.0     | 25.0 | 20     | 200     |     |         |  |
|         | 123456 | MONPNT3  | FS_1000 | 123  | -4.482 | FS_1000 | 356 | -11.385 |  |
|         |        | AMONPNT1   | ROOT    | 123  | 4.482  | R00T    | 456 | 11.385  |  |

**Alternate Format:**

| 1       | 2     | 3        | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|-------|----------|--------|--------|--------|--------|--------|--------|----|
| MONSUM1 | NAME  | LABEL    |        |        |        |        |        |        |    |
|         | CP    | X        | Y      | Z      | CD     |        |        |        |    |
|         | MTYPE | NEWAXISA | NAME1a | AXES1A | COEF1A | NAME2A | AXES2A | COEF2A |    |
|         |       |          | NAME3A | AXES3A | COEF3A | Etc    |        |        |    |
|         |       | NEWASIXB | NAME2A | AXES1B | COEF1B | NAME2B | AXIS2B | COEF2B |    |
|         |       |          | NAME3B | AXES3B | COEF3B | Etc.   | .      |        |    |
|         |       | NEWAXISC | Etc    |        |        |        |        |        |    |

**Alternate Example:** Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

|         |          |  |         |      |        |         |     |         |  |
|---------|----------|--|---------|------|--------|---------|-----|---------|--|
| MONSUM1 | ROOT     | SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS |         |      |        |         |     |         |  |
|         |          | 500.   | 0.0     | 25.0 | 20     |         |     |         |  |
|         | MONPNT3  | 123456   | FS_1000 | 123  | -4.482 | FS_1000 | 456 | -11.385 |  |
|         | AMONPNT1 | 123456   | ROOT    | 123  | 4.482  | R00T    | 456 | 11.385  |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Character string of up to 8 characters identifying the monitor result (Character)   |
| LABEL      | A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.   |
| CP         | The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)   |
| X,Y,Z      | The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).   |
| CD         | The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)                                |
| NEWAXESj   | The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 5. and 6.) |
| MTYPi      | Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.  |
| NAMEij     | Name of the monitored quantity that is to be summed.  |
| AXESij     | Component axes that are to be scaled. (any unique combination of the integers 1 to 6 with no embedded blanks. Must be a subset of NEWAXESj).Default =same as NEWAXESj                                       |
| COEFij     | Scaling coefficient to be applied to the component(s) called out on AXESij field. (Real, Default=1.0)   |

#### Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The result of the MONSUM1 entry is to create new monitor point(s) as follows:

$$\text{MONSUM1}_j = \sum_i^n \text{COEF}_{ji} \text{MR}_i$$

Where  $\text{MR}_i$  is the result from the individual component.

3. The merged monitor points must be of a similar type. “Similar” types are defined as:
  - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1's the MTYPij can be aerodynamic or structural. MTYPij=AMONPT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are the same as NEWAXISj.

6. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6. There may be as many as 6 AXISij and COEFij for each referenced NAMEij.
7. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM1 entry.
8. If multiple types are specified on a MONSUM1, the resulting entry is of a type that appears on the MONSUM1 with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
9. A MONSUM1 can reference the results of another MONSUM1 or a MONSUMT as long as there is not a circular reference.
10. The MONSUM1 entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
11. The CP, CD, and X Y Z location coordinates are for reference only and do not affect the calculations.

**MONSUMT**

Linear combination of two or more monitor points with moment transfer

Defines a new monitor result that is the sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3) 3.)

Format:

| 1       | 2     | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|---------|-------|--------|--------|--------|--------|--------|--------|--------|----|
| MONSUMT | NAME  | LABEL  |        |        |        |        |        |        |    |
|         | AXES  | CP     | X      | Y      | Z      | CD     |        |        |    |
|         | MTYP1 | NAME1a | NAME1b | NAME1c | NAME1d | NAME1e | NAME1f | NAME1g |    |
|         | MTYP2 | NAME2a | NAME2b | NAME2c | NAME2d | NAME2e | NAME2f | NAME2g |    |

Example: Create a new monitor point result by adding an aerodynamic and a monpnt3

|         |          |                      |          |         |         |    |  |  |
|---------|----------|----------------------|----------|---------|---------|----|--|--|
| MONSUMT | FS_900   | FUSELAGE STATION 900 |          |         |         |    |  |  |
|         | 123456   |                      | 500.     | 0.0     | 25.0    | 20 |  |  |
|         | MONPNT3  | FS_1000              | FS_1100  | FS_1200 | MONPNT3 |    |  |  |
|         | AMONPNT1 | HTP_ROOT             | AMONPNT1 |         |         |    |  |  |

| Descriptor | Meaning   |
|------------|---|
| NAME       | Character string of up to 8 characters identifying the monitor result (Character)   |
| LABEL      | A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.   |
| CP         | The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)   |
| X,Y,Z      | The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).   |
| CD         | The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)          |
| AXES       | The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks). |
| MTYPi      | Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.  |
| NAMEij     | Name of the monitored quantity that is to be summed.  |

Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The NAME must be unique with respect to all monitor quantities. The result of the MONSUMT entry is to create new monitor point(s) as follows:

$$\text{MONSUMT}_j = \sum_i^n T_{ji} \text{MR}_i$$

Where  $\text{MR}_i$  is the result from the monitor points being summed and  $T_{ji}$  is set of partial rigid body vectors for the locations of the monitor points being summed, with the origin at the X, Y, Z location.

3. The summed monitor points must be of a similar type. "Similar" types are defined as:
  - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1's the MTYPij can be aerodynamic or structural. MTYPij=AMONPNT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple types are specified on a MONSUMT, the resulting entry is of a type that appears on the MONSUMT with the following order of precedence: If multiple types are specified on a MONSUM1, the resulting entry is of a type that appears on the MONSUM1 with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
6. A MONSUMT can reference the results of another MONSUMT or a MONSUM1 as long as there is not a circular reference.
7. The MONSUMT entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).

**MPC****Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where  $u_j$  represents degree-of-freedom Cj at grid or scalar point Gj.

Format:

| 1   | 2   | 3  | 4  | 5  | 6      | 7  | 8  | 9 | 10 |
|-----|-----|----|----|----|--------|----|----|---|----|
| MPC | SID | G1 | C1 | A1 | G2     | C2 | A2 |   |    |
|     |     | G3 | C3 | A3 | -etc.- |    |    |   |    |

Example:

|     |   |    |   |       |   |  |      |  |  |
|-----|---|----|---|-------|---|--|------|--|--|
| MPC | 3 | 28 | 3 | 6.2   | 2 |  | 4.29 |  |  |
|     |   | 1  | 4 | -2.91 |   |  |      |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| Gj         | Identification number of grid or scalar point. (Integer > 0)   |
| Cj         | Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.) |
| Aj         | Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)  |

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The first degree-of-freedom (G1, C1) in the sequence is defined to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.
3. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
4. The m-set degrees-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See the “Degree-of-Freedom Sets” on page 1557 for a list of these entries.
5. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “Parameters” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

**MPCADD****Multipoint Constraint Set Combination**

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

**Format:**

| 1      | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|--------|----|----|----|----|----|----|
| MPCADD | SID | S1 | S2     | S3 | S4 | S5 | S6 | S7 |    |
|        | S8  | S9 | -etc.- |    |    |    |    |    |    |

**Example:**

|        |     |   |   |   |   |   |  |  |  |
|--------|-----|---|---|---|---|---|--|--|--|
| MPCADD | 101 | 2 | 3 | 1 | 6 | 4 |  |  |  |
|--------|-----|---|---|---|---|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)  |
| Sj         | Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0) |

**Remarks:**

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.
3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.
4. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “[Parameters](#)” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,101.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**MPCAX****Conical Shell Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

for conical shell coordinates, where  $u_j$  represents the degree-of-freedom  $C_j$  at ring RID $j$  and harmonic HID $j$ .

Format:

| 1     | 2    | 3    | 4  | 5  | 6      | 7    | 8  | 9  | 10 |
|-------|------|------|----|----|--------|------|----|----|----|
| MPCAX | SID  |      |    |    | RID1   | HID1 | C1 | A1 |    |
|       | RID2 | HID2 | C2 | A2 | -etc.- |      |    |    |    |

Example:

|       |    |   |   |      |    |   |   |     |  |
|-------|----|---|---|------|----|---|---|-----|--|
| MPCAX | 32 |   |   |      | 17 | 6 | 1 | 1.0 |  |
|       | 23 | 4 | 2 | -6.8 |    |   |   |     |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0)                      |
| RID $j$    | Ring identification number. (Integer > 0)                     |
| HID $j$    | Harmonic identification number. (Integer > 0)                 |
| C $j$      | Component number. (1 ≤ Integer ≤ 6)                           |
| A $j$      | Coefficient. (Real; Default = 0.0 except A1 must be nonzero.) |

Remarks:

1. MPCAX is allowed only if an AXIC entry is also present.
2. The first degree-of-freedom in the sequence is assumed to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPCAX entry cannot be assigned dependent by another MPCAX entry.
3. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
4. Dependent degrees-of-freedom appearing on MPCAX entries may not appear on OMITAX, SPCAX, or SUPAX entries.
5. See [Conical Shell Element \(RINGAX\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion of the problem.
6. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in “[Parameters](#)” on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,32.

**MPCD****Load Selectable Value for  $Y_m$  of Non Homogenous Multipoint Constraint**

Defines a load selectable value for  $Y_m$  of a MPCY entry.

Format:

| 1    | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9 | 10 |
|------|-----|-----|-----|-----|-----|-----|-----|---|----|
| MPCD | SID | GM1 | CM1 | YM1 | GM2 | CM2 | YM2 |   |    |

Example:

|      |     |     |   |     |  |  |  |  |  |
|------|-----|-----|---|-----|--|--|--|--|--|
| MPCD | 700 | 101 | 2 | .06 |  |  |  |  |  |
|------|-----|-----|---|-----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number of the MPCD entry. (Integer > 0)  |
| GMi        | Grid or scalar point identification number. Along with Ci it identifies the Equation of a MPCY. (Integer > 0) |
| Ci         | Component number. (Any one of the Integers 1 through 6 for grid points, blank, zero or 1 for scalar points.)  |
| YMi        | Right hand side value of MPC equation. (Real)   |

Remarks:

1. In static solution sequences, the SID is selected by the LOAD Case Control command.
2. The Gi,Ci referenced on this entry must also be referenced on a MPCY Bulk Data entry and selected by an MPC Case Control command.
3. Values YMi will override the value specified on an MPCY Bulk Data entry.
4. The LOAD Bulk Data entry will not combine an MPCD load entry.
5. Two separate MPC equations may be pointed to per entry.

**MPCREEP****SOL 600 Advanced Creep Options**

Specifies input values for Marc's creep parameter when creep analysis is performed using SOL 600 only.

**Format:**

| 1       | 2     | 3     | 4      | 5    | 6 | 7 | 8 | 9 | 10 |
|---------|-------|-------|--------|------|---|---|---|---|----|
| MPCREEP | ITYPE | IMPLK | IIMPLM | IHOW |   |   |   |   |    |

**Example:**

|         |   |   |   |   |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|
| MPCREEP | 1 | 0 | 0 | 0 |  |  |  |  |  |
|---------|---|---|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ITYPE      | Type of explicit creep analysis enter one of the following: (Integer; Default = 0)                                  |
| 0          | Normal creep (Maxwell Model)  |
| 1          | Viscoplastic creep  |
| 2          | Viscoplastic creep with nonassociative flow rule.   |
| IMPLK      | Flag determining if the explicit Kelvin model is to be used. (Integer; Default = 0)                                 |
| 0          | Explicit Kelvin model is not used   |
| 1          | Explicit Kelvin model is used.  |
| IIMPLM     | Flag determining if the implicit Maxwell creep or implicit viscoplastic model is to be used. (Integer; Default = 0) |
| 0          | These models are not used.  |
| 1          | The implicit Maxwell creep or implicit viscoplastic model is used.  |
| IHOW       | Flag specifying how the Maxwell creep model or implicit viscoplastic model is used. (Integer; Default = 0)          |
| 0          | Use elastic tangent   |
| 1          | Use secant tangent  |
| 2          | Use radial return   |

**Remarks:**

1. Set IHOW=0 or leave field blank if IIMPLM=0.
2. When using the implicit Maxwell creep model, the stress dependence must be in exponential form and the CRPLAW user subroutine cannot be used.
3. Only one MPCREEP entry may be entered in the input. If additional MPCREEP entries are found, the first will be used.
4. If ITYPE > 0, Bulk Data entry, MACREEP or MTCREEP is required in addition to this entry.

5. This entry maps to Marc's CREEP parameter.
6. This entry is not necessary if fields 2-5 are all zero or blank.
7. Kelvin creep requires user subroutines.

**MPCY****Non Homogenous Multipoint Constraint**

Defines a multipoint constraint equation of the form

$$A_m u_m + \sum_i A_i u_i = Y_m$$

where  $u_m$  represents degree-of-freedom  $C_m$  at grid or scalar point  $G_m$  defined to be the dependent degree-of-freedom.  $u_i$  represents degree-of-freedom  $C_i$  at grid or scalar point  $G_i$  defined to be the independent degree-of-freedom.  $Y_m$  is a value for the equation.

**Format:**

| 1    | 2   | 3  | 4  | 5  | 6   | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|-----|----|----|---|----|
| MPCY | SID | GM | CM | AM | YM  |    |    |   |    |
|      |     | G1 | C1 | A1 | G2  | C2 | A2 |   |    |
|      |     | G3 | C3 | A3 | ... |    |    |   |    |

**Example:**

|      |    |      |   |     |       |  |  |  |  |
|------|----|------|---|-----|-------|--|--|--|--|
| MPCY | 70 | 205  | 1 | 1.0 | .02-3 |  |  |  |  |
|      |    | 1608 | 1 | 1.2 |       |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number of a multi-point constraint set. (Integer > 0)                                     |
| GM         | Identification number of grid or scalar point. (Integer > 0)   |
| Cj         | Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.) |
| AM         | Coefficient. (Real nonzero value)  |
| YM         | Right hand side value. (Real; Default = 0.0)   |
| Gi         | Identification number of grid or scalar point. (Integer > 0)   |
| Ci         | Component number. (Any one of the Integers 1 through 6 for grid point; blank or zero for scalar points.)     |
| Ai         | Coefficient. (Real; Default = 0.0)   |

**Remarks:**

1. Multipoint constraint sets must be selected with Case Control command, MPC=SID.
2. The m-set degree-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. The GM term is an equation marker. If PARAM,AUTOMSET,YES is on Nastran may choose internally another Gi as the actual dependent degree-of-freedom.

**MPHEAT**

Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis

Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.

Format:

| 1      | 2     | 3     | 4     | 5      | 6     | 7    | 8    | 9 | 10 |
|--------|-------|-------|-------|--------|-------|------|------|---|----|
| MPHEAT | IDIST | LHMAX | INTPT | ICONVT | ILAYC | LINR | NONH |   |    |

Example:

|        |   |   |   |   |   |   |   |  |  |
|--------|---|---|---|---|---|---|---|--|--|
| MPHEAT | 1 | 1 | 1 | 0 | 3 | 1 | 0 |  |  |
|--------|---|---|---|---|---|---|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDIST      | Temperature distribution in thickness direction of heat transfer shell elements. (Integer; Default = 0)    |
| 0          | Linear Variation (membrane)  |
| 1          | Quadratic variation  |
| LHMAX      | Maximum number of latent heats associated with any material. (Integer; Default = 0)                        |
| INTPT      | Controls integration point output. (Integer; Default = 0)  |
| 0          | No integration point output  |
| 1          | Gradients and fluxes at integration points   |
| 2          | Save as 1 plus flux values at grid points  |
| ICONVT     | Flag to include convective terms. (Integer; Default = 0)   |
| 0          | Do not include convective terms  |
| 2          | Include convective terms   |
| LAYC       | Number of through thickness layers for heat transfer shells. (Integer; Default = 1)                        |
| LINR       | Flag controlling linearization of surface energy and receding surface calculations. (Integer; Default = 1) |
| 1          | Linearize the calculations   |
| 2          | Do not linearize the calculations  |
| NONH       | Flag to store nonhomogeneous density for postprocessing. (Integer; Default = 0)                            |
| 0          | Do not store the values  |
| 1          | Store the values   |

**MPROCS****Defines a Processor Entry to be Used in the SOL 600 Marc Parameter Section**

Defines a processor entry to be used in the SOL 600 Marc Parameter Section. It controls the use of vectorization and parallelization in the element assembly phase in Marc. Used in SOL 600 only.

**Format:**

| 1      | 2   | 3    | 4     | 5    | 6  | 7  | 8 | 9 | 10 |
|--------|-----|------|-------|------|----|----|---|---|----|
| MPROCS | I2  | I3   | I4    | I5   | I6 | I7 |   |   |    |
|        | IRF | IFGF | IAXIS | WGHT |    |    |   |   |    |
|        | Dx  | Dy   | Dz    | X    | Y  | Z  |   |   |    |

**Example:**

|        |   |   |   |   |    |  |  |  |  |
|--------|---|---|---|---|----|--|--|--|--|
| MPROCS | 1 | 1 | 1 | 1 | 11 |  |  |  |  |
|--------|---|---|---|---|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| I2         | Value of Marc's 2nd processor field. Number of CPUs to use. (Integer; Default = 0)  |
| I3         | Value of Marc's 3rd processor field. Optimal vector length (Defaults to 32 or 64 depending on the computer system). (Integer; Default = 0)  |
| I4         | Value of Marc's 4th processor field (enter 1 if beta matrices are to be formed in parallel). (Integer; Default = 0)   |
| I5         | Value of Marc's 6th processor field (enter 1 to use DDM single file input). (Integer; Default = 0)  |
| I6         | Value of Marc's 7th processor field -- Domain Decomposition Method. (Integer; Default = 0)<br>Enter 11 to use Metis Best (best method of 12, 13 or 14) decomposition; Default.<br>Enter 12 to use Metis Element-Based decomposition.<br>Enter 13 to use Metis Node-Based decomposition.<br>Enter 14 to use Vector decomposition.<br>Enter 15 to use Radial decomposition.<br>Enter 16 to use Angular decomposition. |
| I7         | Value of Marc's 8th processor field. Enter 0 to use in-core storage for DDM single file creation. Enter 1 to use out-of-core storage. (Integer; Default = 0)  |
| IRF        | Island removal flag for domain decomposition. (Default = 0)<br>0      Do not remove islands<br>1      Attempt to remove islands   |
| IFGF       | Fine graph flag for domain decomposition. (Default = 0)<br>0      Coarse graph<br>1      Fine graph   |

| Descriptor | Meaning   |
|------------|---|
| IAXIS      | Control of point on axis of rotation for radial/angular domain decomposition  |
| 0          | Use centroid of the boundary box of the model   |
| 1          | User supplied point (supply X, Y, Z below)  |
| WGHT       | Element coefficient weight. Controls balance between computational costs of domains, range is 0.0 to 1.0. (Default = 1.0; Use full element weight)<br>0.0 - Do not use element weight |
| Dx         | First direction cosine of vector used for decomposition method 14, 15, or 16  |
| Dy         | Second direction cosine of vector used for decomposition method 14, 15, or 16   |
| Dz         | Third direction cosine of vector used for decomposition method 14, 15, or 16  |
| X          | X coordinate of point on axis   |
| Y          | Y coordinate of point on axis   |
| Z          | Z coordinate of point on axis   |

#### Remarks:

1. This entry should only be made for special cases when using DDM with Marc's single file parallel capability. Do not enter except for SOL 600 parallel executions.
2. Consult the Marc documentation volumes A, B, C for more detailed descriptions of this entry.
3. Enter only one of MPROCS entry in any given file. If more than one is entered, the first encountered will be used.
4. The continuation lines may be omitted if not required.
5. I2, I3, and I4 are no longer used and should be zero or blank.

**MREVERS**

Defines Which Elements Require Node Numbering to be Reversed - SOL 600

This entry is only used if the checks built into SOL 600 somehow fail to predict some elements which need node numbering reversed. Nastran allows clockwise as well as counter clockwise node numbering. Marc only allows counter clockwise numbering for most elements. SOL 600 has builtin logic to check for node reversal, however there may be some instances where Marc gives an “inside out” message during increment (before any nonlinear loads have been applied). This rarely happens unless field 7 of the GRID entry is set, in which case it sometimes happens. When this happens, the user can apply this entry to instruct SOL 600 how to renumber elements which have “inside out” messages during increment zero. (If “inside out” messages occur after increment zero, this is normally due to large loads and the element has either failed or the model needs to be re-meshed).

**Format:**

| 1       | 2   | 3    | 4    | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|-----|------|------|-----|-----|-----|-----|-----|----|
| MREVERS | N1  | M1   | N2   | M2  | N3  | M3  | N4  | M4  |    |
|         | ID1 | ID2  | ID3  | ID4 | ID5 | ID6 | ID7 | ID8 |    |
|         | ID9 | ID10 | etc. |     |     |     |     |     |    |

**Example:**

|         |     |     |     |     |     |  |  |  |  |
|---------|-----|-----|-----|-----|-----|--|--|--|--|
| MREVERS |     | 2   | 3   |     |     |  |  |  |  |
|         | 500 | 510 | 520 | 888 | 889 |  |  |  |  |
|         | 950 |     |     |     |     |  |  |  |  |

**Descriptor      Meaning**

- Ni                  Node number location to be reversed. (Integer; no Default; see Remark 1.)
- Mi                  Paired node number location to be reversed. (Integer; no Default; see Remark 1.)
- IDi                Element identification number. (Integer; no Default)

**Remarks:**

1. Example of node number locations are as follows:  
For 4-node quads the node number locations are 1, 2, 3, 4  
For 8-node hexas the node number locations are 1, 2, 3, 4, 5, 6, 7, 8
2. The 2<sup>nd</sup> and succeeding lines may be used to make it easy to enter elements with “inside out” messages in a preliminary marc.out file. The user can grep for “inside out”, save the message in a file, easily edit the file to retain only the element numbers and then add these to lines 2 and subsequent of the alternate format.
3. Up to four pairs of node locations may be reversed per entry. If additional pairs are necessary, repeat the entry and its continuation lines.
4. The example will reverse nodes locations 2 and 3 for elements 500, 510, 520, 888, 889 and 950.

5. This entry may only be used in MD Nastran R2.1 and later versions.
6. See PARAM,MAXIREVV for a similar option.

**MRSSCON**

Auxiliary data for RSSCON for use in SOL 600

Defines auxiliary data for RSSCON for use in SOL 600. Used in SOL 600 only.

**Format:**

| 1       | 2     | 3   | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|-------|-----|-----|-----|---|---|---|---|----|
| MRSSCON | ICONV | IDB | IDP | IDM | A | J | I | E |    |

**Example:**

|         |   |      |      |     |      |      |      |       |  |
|---------|---|------|------|-----|------|------|------|-------|--|
| MRSSCON | 0 | 5000 | 5000 | 900 | 20.0 | 200. | 100. | 30.0E |  |
|---------|---|------|------|-----|------|------|------|-------|--|

| Descriptor | Meaning   |
|------------|---|
| ICONV      | Flag indicating how RSSCON rigid elements will be converted in SOL 600. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0 Convert all RSSCON to CBEAM using values in fields 2-8</li> <li>1 Convert all RSSCON to RBE2 using 3 degrees of freedom</li> <li>2 Convert all RSSCON to RBE2 using 6 degrees of freedom</li> </ul> |
| IDB        | Starting CBEAM ID for all RSSCON's that are mapped to CBEAM's or RBE2 ID if RSSCON's are mapped to RBE2's (Integer; Default = 99000; see Remark 1.)   |
| IDP        | PBEAM ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 5. (Integer; Default = 99000)   |
| IDM        | MAT1 ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 6. (Integer; Default = 99000)  |
| A          | Area to be placed in the PBEAM (Real; Default = 10.0)   |
| J          | Torsional moment of inertia to be placed in the PBEAM (Real; Default = 20.0)  |
| I          | Moment of inertia (in both directions) to be placed in the PBEAM (Real; Default = 10.0)   |
| E          | Elastic modulus to be placed in the IDM MAT1 entry (Real; Default = 1.0E6)  |

**Remarks:**

1. ICONV=1 or 2 may cause MPC conflicts and/or may fail if AUTOMSET,YES is used.
2. RSSCON type=elem is not presently supported by SOL 600.
3. This entry applies to RSSCON type=grid where ES1, EA1, EB1 all have different coordinates, and if used ES2, EA2 and EB2 do not have the same coordinates as each other (see RSSCON entry for meaning of these values).
4. Only one MRSSCON entry should be include in the input data. If more than one is included, the first as determined by XSORT, will be used.

5. If IDP is entered, a PBEAM with and id of IDP and properties A, J, and I will automatically be created.
6. If IDM is entered, a MAT1 with a Young's modulus of E and Poisson ratio of 0.3 will automatically be created.

2550

**MSTACK**

Defines the Direction in Which 3D Solid Composites are Stacked -SOL 600

**MSTACK**

Defines the Direction in Which 3D Solid Composites are Stacked -SOL 600

Defines the direction in which 3D Solid Composites are stacked. Used in SOL 600 only.

**Format:**

| 1      | 2   | 3    | 4   | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|-----|------|-----|------|---|---|---|---|----|
| MSTACK | ID1 | THRU | ID2 | IDIR |   |   |   |   |    |

**Example:**

|        |     |      |     |   |  |  |  |  |  |
|--------|-----|------|-----|---|--|--|--|--|--|
| MSTACK | 100 | THRU | 200 | 1 |  |  |  |  |  |
|--------|-----|------|-----|---|--|--|--|--|--|

**Descriptor      Meaning**

- |      |   |
|------|---|
| ID1  | Starting solid element ID (Integer; Required; no Default)   |
| THRU | Enter the character string THRU if more than one element is desired   |
| ID2  | Ending solid element in a range of ID1 to ID2 (Integer; optional; no Default)   |
| IDIR | Stacking direction for this range of solid elements (Integer; Required; Default = 1) <ul style="list-style-type: none"> <li>1      Thickness direction is 1-2-3-4 face to 5-6-7-8 face (for CHEXA)</li> <li>2      Thickness direction is 1-5-8-4 face to 2-6-7-3 face (for CHEXA)</li> <li>3      Thickness direction is 2-6-5-1 face to 3-7-8-4 face (for CHEXA)</li> </ul> |

**Remarks:**

1. Enter as many MSTACK lines as necessary to define any solid composites where the defaults do not apply. If IDIR is 1 for all elements in the range, this entry is not required.
2. This entry equates to Marc's EGEOM3 value for solid composite elements (for example element 149).

**MT16SEL**

Limits results to selected elements or grids for t16 and t19 for SOL 600

Limits elements and/or grid results to selected elements or grids for t16 and t19 file results. Used in SOL 600.

**Format**

| 1       | 2    | 3   | 4    | 5   | 6  | 7   | 8 | 9 | 10 |
|---------|------|-----|------|-----|----|-----|---|---|----|
| MT16SEL | TYPE | ID1 | THRU | ID2 | BY | ID3 |   |   |    |
|         | TYPE | ID1 | THRU | ID2 | BY | ID3 |   |   |    |

**Example:**

|         |      |      |      |      |    |    |  |  |  |
|---------|------|------|------|------|----|----|--|--|--|
| MT16SEL | GRID | 1    | THRU | 100  |    |    |  |  |  |
|         | GRID | 2525 | THRU | 3000 | BY | 10 |  |  |  |
|         | ELEM | 100  | THRU | 500  | BY | 2  |  |  |  |
|         | ELEM | 1000 | THRU | 2000 |    |    |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| TYPE       | Type of output - Enter one of the values "GRID", or "ELEM", for "GRID", nodal results such as displacement, spc force, etc are selected. For "ELEM", element results such as stress, strain, etc. are selected. This entry should normally be used in combination with MARCOUT unless the MARCOUT defaults are satisfactory for the model. |
| ID1        | Starting ID of above selection. (Integer; no Default required value)   |
| ID2        | Ending ID of above selection. (Integer; Default = ID1)   |
| ID3        | Increment by value. (Integer; Default = 1)   |

**Remarks:**

1. It is highly recommended that all grid and element output be placed on the t16 file since it is not usually known where the maximum values will occur and the max/min values could easily be missed. Also, if all grid/element values are not selected, contour plots could be misleading. To output all grid/element results in the t16 file, do not make any MT16SEL entries.
2. This entry may not be used when OUTR options (which requires a t16 to op2 conversion) on the SOL 600 entry are requested. If this entry is made together with any OUTR options, this entry will be ignored and a warning message issued.
3. ID2 must be blank, zero or greater than ID1 (if ID2 is blank or zero, only ID1 will be used)
4. ID3 must not be negative. If can be blank or zero in which case it is reset to one.
5. This entry (with the exception of Remark 3) activates Marc's POST version 13 and overrides any other POST version specifications such as PARAM,MARCPOST.
6. This entry can be used to also obtain a t19 file with selected element and/or grid results by placing PARAM,MARCT19,1 in the bulk data.

2552

**MT16SEL**

Limits results to selected elements or grids for t16 and t19 for SOL 600

7. It is not presently possible to only create a t19 file with selected element/grid results using SOL 600.  
If a t19 file is desired, a t16 file must also be created.

**MT16SPL**

Split a Marc t16 file into one or more smaller t16 files in SOL 600

Determines how to split a Marc t16 file into one or more smaller t16 files in SOL 600. Splitting of a large t16 file is sometimes necessary if the postprocessor aborts due to the large amount of data or if the results need to be transferred to another computer for postprocessing.

Format: This entry is available in small field format only:

| 1       | 2    | 3     | 4     | 5    | 6          | 7 | 8 | 9 | 10 |
|---------|------|-------|-------|------|------------|---|---|---|----|
| MT16SPL | IOPT | NVECT | FNAME |      |            |   |   |   |    |
|         | ID1  | START | LAST  | INCR | (remark 4) |   |   |   |    |
|         | ID2  | START | LAST  | INCR | LastFew    |   |   |   |    |

Examples:

|         |   |   |     |   |   |  |  |  |  |
|---------|---|---|-----|---|---|--|--|--|--|
| MT16SPL | 0 |   |     |   |   |  |  |  |  |
|         | 1 | 5 | 100 | 5 | 3 |  |  |  |  |

|         |   |    |          |       |   |  |  |  |  |
|---------|---|----|----------|-------|---|--|--|--|--|
| MT16SPL | 0 |    | old1.mar | c.t16 |   |  |  |  |  |
|         | 1 | 10 | 80       | 10    | 3 |  |  |  |  |
|         | 2 | 90 | 100      | 5     | 4 |  |  |  |  |

|         |   |     |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|
| MT16SPL | 1 | 1   |  |  |  |  |  |  |  |
|         | 1 | 100 |  |  |  |  |  |  |  |

|         |   |    |          |    |   |  |  |  |  |
|---------|---|----|----------|----|---|--|--|--|--|
| MT16SPL | 3 | 2  | old2.t16 |    |   |  |  |  |  |
|         | 1 | 10 | 100      | 10 | 3 |  |  |  |  |

Create several new t16 files with 2 increments each using original increments 10, 20, ... 98, 99, 100. This is a restart job and uses old2.t16 (from a standalone Marc run). New files will be created as follows:

jid.0001.t16 with old increments 10, 20  
 jid.0002.t16 with old increments 30, 40  
 jid.0003.t16 with old increments 50, 60  
 jid.0004.t16 with old increments 70, 80  
 jid.0005.t16 with old increments 90, 98  
 jid.0006.t16 with old increments 99, 100

| Descriptor | Meaning   |
|------------|---|
| IOPT       | Option of how to split up the “old” t16 file. (Integer; Default = 0)  |
| 0          | Split into as many new t16 files as there are continuation lines of this entry, use the current jid.marc.t16 file, FNAME should not be entered. |
| 1          | Split into as many new t16 files with NVECT original increments each, use the current jid.marc.t16 file, FNAME should not be entered.           |

| Descriptor | Meaning   |
|------------|---|
| 2          | Split into as many new t16 files as there are continuation lines of this entry, use a previously generated t16 file whose name is specified by FNAME.   |
| 3          | Split into as many new t16 files with NVECT original increments each, use a previously generated t16 file whose name is specified by FNAME.   |
| NVECT      | Number of increments to be placed on each new t16 file (only used if IOPT is 1 or 3)<br>(Integer; no Default)   |
| FNAME      | Original t16 filename - Only used if IOPT is 2 or 3. (Character; no Default) The filename may extend from fields 4-9. If it is more than 8 characters long, this entry must made in fixed format. the entire t16 filename including the t16 extension should be entered. No upper case letters may be used. (Integer > 0) |
| IDi        | New t16 plot ID (must start with 1 and increase by 1 on each continuation line.)<br>(Integer; no Default)   |
| START      | Starting Marc increment to be placed on new t16 file. (Integer; Default = 1)  |
| LAST       | Last Marc increment in range of start-last-incr to be n new t16 file. (Integer; Default is last increment on original t16 file if FNAME is blank. If FNAME is not blank, LAST must be an accurate value which can be obtained from the .sts file of the original run that produced FNAME.)                                |
| INCR       | Increments to be used for start-last-incr tgo b e on new t16 file. (Integer; Default = 1)   |
| Last Few   | In addition to START,LAST,INCR the last several increments may be placed on the t16 file. In the first example increments 5, 10, 15, ..., 100, 99, 98 are placed on the new t16 file. LastFew may only be entered on the last line. (Integer; Default = 2)  |

### Remarks:

1. Sometimes large SOL 600 models do not converge on a user wishes to examine output at unknown time intervals. To determine what is happening, it is frequently necessary to plot the results at several output intervals. In fact, sometimes the last increment may have bogus results due to divergence. At present, MSC GUI programs sometimes are not capable of postprocessing the large amount of data one would like to include in a single f16 file. This option allows you to break up the t16 file into one or more smaller files.
2. GUI's might only be able to handle one increment per t16 file for extremely large models. To specify this, only enter Id and START as in the 3rd example.
3. All t16 files will have the geometry as well as the specified output increments.
4. LastFew may only be entered on the last line.
5. Nastran may be restarted to perform this step. To do so, enter  
SOL 600, ID t16split=fname  
Where fname is the jid of the original job.
6. The new files will be named jid.ID.t16. Examples are as follows:

Case 1 - Split up t16 as part of current run starting with jid1.dat as the Nastran input SOL 600 will create jid1.marc.t16 with a full set of output increments (unless reduced by you).

jid1.0001.t16  
jid1.0002.t16  
etc.

Case 2 - Split up a t16 file formed by a previous SOL 600 run named jid1.dat (the t16 file is named jid1.marc.t16). The current Nastran input file to split up the original t16 is named jid2.dat. The new t16 files will be designated:

jid2.00001.t16  
jid2.00002.t16  
etc.

7. Nastran will spawn Marc's pldump2000 program to split up the original t16 file.
8. If FNAME is entered (IOPT=2 or 3), OUTR options (on SOL600,ID) will be ignored. If FNAME is blank and IOPT=0 or 1 and OUTR options are requested, the t16op2 translator process the full (unsplit) jjid.mar.t15 file. Future implementations may allow processing of the split t16 files.
9. Only one MT16SPL (plus many continuation lines as necessary) is allowed per job.

**MTABRV**

Defines a List of Tables to Modify and/or Positions of Values - SOL 600

In some cases, a model is built with tables defined backwards. For example, for a stress-strain curve the compression position of the curve might really need to be the tension portion and visa versa. This entry allows them to be reversed. In addition, this entry allows a user to add a point to the lower and/or upper end of the tables in SOL 600 only.

**Format:**

| 1      | 2   | 3    | 4   | 5    | 6    | 7    | 8     | 9     | 10 |
|--------|-----|------|-----|------|------|------|-------|-------|----|
| MTABRV | ID1 | THRU | ID2 | IREV | XLOW | YLOW | XHIGH | YHIGH |    |

**Example:**

|        |    |      |    |   |       |        |      |        |  |
|--------|----|------|----|---|-------|--------|------|--------|--|
| MTABRV | 1  | THRU | 10 | 1 | -100. | -1.0E7 | 0.01 | 1.0E7  |  |
|        | 21 | THRU | 25 | 1 |       |        |      |        |  |
|        | 45 | THRU | 48 |   |       |        | 25.5 | 66523. |  |

| Descriptor | Meaning  |   |                |   |         |
|------------|--|---|----------------|---|---------|
| ID1        | Starting table ID to be reversed. (Integer; Required; no Default)  |   |                |   |         |
| THRU       | Enter the character string THRU if more than one grid is desired.  |   |                |   |         |
| ID2        | Ending table ID of the range. (Integer; or blank. If blank; ID2=ID1)   |   |                |   |         |
| IREV       | Option to reverse (flip) positive and negative values and change the sign for tables in the range ID1 to ID2. (Integer; Default = 0) <table border="0" style="margin-left: 20px;"> <tr> <td>0</td> <td>Do not reverse</td> </tr> <tr> <td>1</td> <td>Reverse</td> </tr> </table> | 0 | Do not reverse | 1 | Reverse |
| 0          | Do not reverse   |   |                |   |         |
| 1          | Reverse  |   |                |   |         |
| XLOW       | Option to add a point to the lower end of these tables. If so, XNEG is the lower end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)  |   |                |   |         |
| YLOW       | Option to add a point to the lower end of these tables. If so, YNEG is the lower end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)  |   |                |   |         |
| XHIGH      | Option to add a point to the upper end of these tables. If so, XPOS is the upper end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)  |   |                |   |         |
| YHIGH      | Option to add a point to the upper end of these tables. If so, YPOS is the upper end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)  |   |                |   |         |

**Remarks:**

- Missing tables in the range ID1 to ID2 will be ignored and no error will be produced.
- This capability is limited to tables defined using TABLES1, TABLED1 and/or TABLEM1 and the behavior in the x and y directions must be linear.

3. Values XLOW, YLOS, XHIGH, YHIGH are not reversed and the signs are not changed even if IREV=1.
4. The first example reverses tables 1-10 and also adds a point at the lower and upper end. The second example reverses tables 21-25, no additional points are added. The third example adds a point to the upper end of tables 45-48, does not reverse the tables or add a point to the lower end.
5. This entry will reverse both X and Y of the specified tables and is not capable of reversing only X or Y.

**MTCREEP****AUTO THERM CREEP Iteration Control - SOL 600**

Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600 only.

**Format:**

| 1       | 2     | 3    | 4    | 5     | 6    | 7     | 8     | 9    | 10 |
|---------|-------|------|------|-------|------|-------|-------|------|----|
| MTCREEP | ID    | Tchg | Nmax | Iasmb | Ttot | Tincr | Tcrep | Nsub |    |
|         | Maxit | Nupd | VV1  | VV2   | VV3  | IABS  |       |      |    |

**Example:**

|         |    |     |       |    |       |     |       |    |  |
|---------|----|-----|-------|----|-------|-----|-------|----|--|
| MTCREEP | 11 | 1.0 | 99999 | 1  | 2000. | 10. | 1000. | 50 |  |
|         | 5  | 1   | .5    | .1 | .05   |     |       |    |  |

| Descriptor | Meaning  |   |
|------------|--|---|
| ID         | Identification number of a matching NLPARM Case Control command (for statics) or entry (for dynamics). (Integer; no Default; Required field) |   |
| Tchg       | Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)                              |   |
| Nmax       | Maximum number of increments allowed. (Integer; Default = 999999) (2,2)  |   |
| Iasmb      | Reassembly interval for element matrices. (Integer; Default = 1) (2,3)   |   |
| Ttot       | Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)   |   |
| Tincr      | Suggested time increment for creep analysis (Real; no Default) (3,1)   |   |
| Tcrep      | Total creep time to be covered in this creep analysis (Real; no Default) (3,2)   |   |
| Nsub       | Maximum number of subincrements to be allowed during this creep analysis (Integer; Default = 50). (3,3)                                      |   |
| Maxit      | Maximum number of iterations allowed to modify the time step during any increment. (Integer; Default = 5) (3,4)                              |   |
| Nupd       | Number of increments between stiffness matrix updates. (Integer; Default = 1) (3,5)  |   |
| VV1        | Tolerance value #1. (Real; see below for defaults) (4,1)   |   |
|            | IABS=0   | Enter the tolerance on the creep strain increment relative to the elastic strain. (Default = 0.5)   |
|            | IABS=1   | Enter the maximum creep strain increment. (Default = 0.01)  |
| VV2        | Tolerance value #2 (Real; see below for defaults) (4,2)  |   |
|            | IABS=0   | Enter the tolerance on the stress change per increment divided by the total stress. (Default = 0.1) |
|            | IABS=1   | Enter the maximum stress increment. (Default = 100.0)   |

| Descriptor | Meaning   |
|------------|---|
| VV3        | Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05) (4,3) |
| IABS       | Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (4,5)   |
| 0          | Relative checking is used   |
| 1          | Absolute checking is used   |

**Remarks:**

1. This entry maps to Marc's AUTO THERM CREEP entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entries, MPCREEP and MCHSTAT, must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO THERM CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

**MTHERM**

## Iteration Control for Automatic Thermal Loading for Structural Analysis Following a SOL 600 Heat Transfer Analysis

Iteration control for automatic thermal loading for structural analysis following a SOL 600 heat transfer analysis.

**Format:**

| 1      | 2  | 3    | 4    | 5     | 6    | 7     | 8 | 9 | 10 |
|--------|----|------|------|-------|------|-------|---|---|----|
| MTHERM | ID | Tchg | Nmax | Iasmb | Ttot | Tincr |   |   |    |

**Example:**

|        |    |     |       |   |       |     |  |  |  |
|--------|----|-----|-------|---|-------|-----|--|--|--|
| MTHERM | 21 | 1.0 | 99999 | 1 | 2000. | 10. |  |  |  |
|--------|----|-----|-------|---|-------|-----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of a matching Case Control NLPARM command (for statics) or entry (for dynamics). (Integer; no Default; Required field) |
| Tchg       | Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)                              |
| Nmax       | Maximum number of increments allowed. (Integer; Default = 999999) (2,2)  |
| Iasmb      | Reassembly interval for element matrices. (Integer; Default = 1) (2,3)   |
| Ttot       | Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)   |
| Tincr      | Suggested time increment for creep analysis. (Real; Default = 0.0; leave blank if analysis is not a creep analysis) (2,5)                    |

**Remarks:**

1. This entry maps to Marc's AUTO THERM entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. (i,j) refer to Marc's AUTO THERM (data block, field).
4. This entry may be omitted in which case an AUTO THERM entry as follows will be used  
AUTO THERM  
1., 9999, 0, 1.0

**NHRMPRM**

## Parameters for Nonlinear frequency response analysis (SOL 128)

Format:

| 1       | 2      | 3    | 4      | 5    | 6 | 7 | 8 | 9 | 10 |
|---------|--------|------|--------|------|---|---|---|---|----|
| NHRMPRM | PARAM1 | VAL1 | PARAM2 | VAL2 |   |   |   |   |    |

Example:

|         |       |   |         |   |  |  |  |  |  |
|---------|-------|---|---------|---|--|--|--|--|--|
| NHRMPRM | MNCSF | 3 | NLHDIAG | 1 |  |  |  |  |  |
|---------|-------|---|---------|---|--|--|--|--|--|

| Var Name | Type/ Default | Description   |
|----------|---------------|---|
| NLHRED   | I = -1        | Use reduction for nonlinear analysis.   |
| NHPLUS   | I = 20        | Number of extra points to avoid aliasing.   |
| NLHTOL   | R = 1.0E-5    | Tolerance for convergence.  |
| NLHTWK   | R = 1.1       | Push-off factor for line search in case convergence fails.  |
| MXICODE0 | I = 5         | If solution fails to converge for mxicode0 steps, new trial displacement is used.   |
| TICPNCH  | I = 0         | If TICPNCH = -1 then write a TIC Bulk Data entry for each degree-of-freedom in the d-set to be included in a subsequent transient analysis run. |
| MNCSF    | I = 1000      | Max number of non-converged sequential frequencies for RESET.   |
| MFRINT   | R = 10.       | Maximum frequency interval for RESET.   |
| NLHDIAG  | I = 0         | Output additional diagnostic messages.  |
| CONTP    | I = 0         | Continuation procedure.   |

Remarks:

- Following parameters can also be specified using PARAM option: NLHRED, NHPLUS, NLHTOL, NLHTWK, TICPNCH and MXICODE0.  
See description under PARAM for more details about these variables.  
In case a parameter is specified in NHRMPRM and also using PARAM, the value specified using PARAM entry is used.
- Parameters MNCSF or MFRINT can be used to reset initial condition in case of non-convergence.  
If both these parameters are specified, the condition which is satisfied first is used.
- CONTP =1 is used to initiate continuation procedure method. In this case, following parameters are not used in the analysis: NLHRED, MNCSF, MFRINT. Currently, the CONTP option is available for frequency-independent analysis only.  
Here, NLHRED is set to 0 to not use the reduction method. And initial condition for the continuation procedure is determined using the results from two previously converged solutions.

**NLADAPT**

## Additional Parameters for Automatic Load or Time Stepping

Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400. *NLADAPT is an obsolete option from MD Nastran 2010 onwards. It is recommended that NLADAPT be replaced by NLSTEP.*

Format:

| 1       | 2       | 3      | 4      | 5      | 6     | 7     | 8      | 9     | 10 |
|---------|---------|--------|--------|--------|-------|-------|--------|-------|----|
| NLADAPT | ID      |        |        |        |       |       |        |       |    |
|         | “STEP”  | RSMALL | RBIG   | TSMIN  | TSMAX | NSMAX | NRECYC | SFACT |    |
|         |         | IDAMP  | DAMP   |        |       |       |        |       |    |
|         | “CREEP” | RAC    | TCSTRN | TCSTRC | TCOFF |       |        |       |    |

Example:

|         |       |   |  |  |  |  |  |  |  |
|---------|-------|---|--|--|--|--|--|--|--|
| NLADAPT | 700   |   |  |  |  |  |  |  |  |
|         | STEP  |   |  |  |  |  |  |  |  |
|         |       | 4 |  |  |  |  |  |  |  |
|         | CREEP | 1 |  |  |  |  |  |  |  |

|         |       |   |  |  |  |  |  |  |  |
|---------|-------|---|--|--|--|--|--|--|--|
| NLADAPT | 800   |   |  |  |  |  |  |  |  |
|         | CREEP | 1 |  |  |  |  |  |  |  |
|         | STEP  |   |  |  |  |  |  |  |  |
|         |       | 4 |  |  |  |  |  |  |  |

**Descriptor**      **Meaning**

|        |   |
|--------|---|
| ID     | Identification number of a NLPARM or entry. (Integer > 0)   |
| “STEP” | Keyword indicating the following entries are for enhanced general load step or time step convergence. (Character) |
| RSMALL | Smallest scale factor for time step changes. See Remark 5. (Real; Default= 0.1)                                   |
| RBIG   | Largest scale factor for time step changes. See Remark 5. (Real; Default = 1.5)                                   |
| TSMIN  | Smallest ratio of a time step to the total time. (Real; Default = total time divided by number of time steps)     |
| TSMAX  | Largest ratio of a time step to the total time. (Real; Default = 0.5)   |
| NSMAX  | Maximum number of steps allowed. (Integer; Default = 99999)   |
| NRECYC | Number of recycles per increment. (Integer; Default = 10)   |
| SFACT  | Scale factor for time step changes. See Remark 6. (Real; Default = 1.2)   |
| IDAMP  | Enter 4 to activate artificial damping. (Integer 0 or 4; Default = 0)   |
| DAMP   | Damping factor for activated artificial damping. (Real; Default = 2.E-4)  |

|         |   |  |
|---------|---|--|
| “CREEP” | Keyword indicating the following entries are for enhanced creep analysis. (Character)   |  |
| RAC     | Flag controlling relative or absolute convergence. (Integer 0 or 1; Default = 0)  |  |
| 0       | Relative checking used.   |  |
| 1       | Absolute checking used.   |  |
| TCSTRN  | Creep strain tolerance. (Real)  |  |
| RAC = 0 | Enter the tolerance on the creep strain increment divided by the elastic strain. (Real; Default = 0.5)  |  |
| RAC = 1 | The maximum creep strain increment allowed. (Real; Default = 0.1)   |  |
| TCSTRS  | Creep stress tolerance (Real)   |  |
| RAC = 0 | Enter the tolerance on the stress increment divided by the total stress. (Real; Default = 0.1)  |  |
| RAC = 1 | enter the maximum stress increment. (Real; Default = 100.0)   |  |
| TCOFF   | Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05) |  |

#### Remarks:

1. This entry must point to an existing NLPARM or entry and is only used in SOL 400.
2. The keywords and their associated entries may occur in any order.
3. This entry computes an initial time step TINIT=1.0/NINC if pointing to a NLPARM entry and TINT=1.0/DT if pointing to a entry.
4. If this entry is used, it is preferred KMETHOD on the NLPARM or METHOD on the be set to PFNT, FNT, ITER, or AUTO.
5. The scale factor is defined as the new step size divided by the time step size.
6. SFACT is only used when artificial damping is activated.

**NLAUTO**

## Parameters for Automatic or Fixed Load/Time Stepping - SOL 600

Defines parameters for automatic or fixed load/time stepping used in SOL 600 only.

**Format:**

| 1      | 2        | 3      | 4      | 5      | 6       | 7      | 8       | 9     | 10 |
|--------|----------|--------|--------|--------|---------|--------|---------|-------|----|
| NLAUTO | ID       | TINIT  | TFINAL | RSMALL | RBIG    | TSMIN  | TSMAX   | NSMAX |    |
|        | NRECYC   | IENHAN | IDAMP  | NSTATE | NCUT    | LIMTAR | IFINISH | FTEMP |    |
|        | SFACT    | IFLAG  | IDTAB  | DAMP   | IDMPFLG |        | IPHYS   | I313  |    |
|        | CRITERIA | SETID  | Y1     | X1     | Y2      | X2     | Y3      | X3    |    |
|        |          | Y4     | X4     |        |         |        |         |       |    |

**Example:**

|        |     |     |   |   |    |      |    |  |  |
|--------|-----|-----|---|---|----|------|----|--|--|
| NLAUTO | 1   | .01 | 1 | 1 | 10 | -1.5 | .5 |  |  |
|        | 5   |     | 1 |   |    | -1   |    |  |  |
|        | 1.2 |     |   |   |    |      |    |  |  |

| Descriptor      | Meaning  |
|-----------------|--|
| ID              | Identification number referenced by the NLPARM, , or TSTEP Case Control command for the applicable subcase. See Remarks 1. and 2. Include a NLPARM or entry for the subcase in addition to the NLAUTO entry. See Remarks 1. and 2. (Integer > 0; no Default) |
| TINIT<br>(2,1)  | Initial time step. (Real; Default is determined by NLPARM NINC or DT, see Remark 8.)   |
| TFINAL<br>(2,2) | Total time period. If OUTR options are used, for a static analysis, TFINAL must be 1.0. (Real; Default is 1.0 or Remark 8.)  |
| RSMALL<br>(2,3) | Smallest ratio between steps. (Real; Default = 0.1 or Remark 7.)   |
| RBIG<br>(2,4)   | Largest ratio between steps. (Real; Default = 10.0 or Remark 7.)   |
| TSMIN<br>(2,5)  | Minimum time step. (Real; Default is total time divided by number of time steps or Remark 7.)  |
| TSMAX<br>(2,6)  | Maximum time step. (Real; Default is total time or Remark 7.)  |
| NSMAX<br>(2,7)  | Maximum number of steps allowed. If IENHAN=2, NSMAX is the number of fixed time steps. (Integer; Default = 999999 or Remark 7.)  |

| Descriptor       | Meaning   |
|------------------|---|
| NRECYC<br>(2,8)  | Desired number of recycles per increment. For beam or shell bending NRECYC should be set to 10. If large deformation is anticipated, NRECYC = NO is necessary for plate and shell buckling. PARAM,MARCD,0 can be used to set NRECYC=10 without entering NLAUTO. (Integer; Default = 5 or Remark 7.)   |
| IENHAN<br>(2,9)  | Enter 1 to activate the enhanced scheme, 0 otherwise. Enter 2 to use fixed time stepping (enter the number of time steps using NSMAX) (Integer)   |
| IDAMP<br>(2,10)  | Artificial damping control. (Integer; Default = 1, or Remark 7.)<br>Enter 0 to not use artificial damping<br>Enter 1 to use artificial damping if time step is less than minima time step. Reduce the minimum time step by 1000.<br>IDAMP=1 is usually necessary (in combination with NRECYC=10) for plate and shell buckling models.<br>Enter 2 to always use artificial damping. The value to use is given by DAMP which scales the damping matrix.<br>Enter 4 to always use artificial damping which is determined by the strain energy in the first increment of the present load case. DAMP is used to scale the strain energy.<br>Enter 5 to set the time step as if artificial damping was being used, but do not actually add artificial damping.<br>Enter 6 to add artificial damping when the minimum time step has been reached. |
| NSTATE<br>(3,1)  | Number of states for post file. (Integer; enter only if ienhan = 1)   |
| NCUT<br>(3,2)    | Maximum number of times to cut down time step in an increment. (Integer) (Default = 10 or Remark 7.; enter only if ienhan = 1)  |
| LIMTAR<br>(3,3)  | Enter 0 to create criteria as limits, 1 to treat criteria as targets. (Integer, Default = 0 or Remark 7.; enter only if ienhan = 1)   |
| IFINISH<br>(3,4) | Enter 1 to finish time period when all nodal temperatures fall below FTEMP. Enter -1 if all nodal temperatures should exceed FTEMP. Enter 0 to omit temperature check. (Integer; enter only if ienhan = 1, Default = 0 or Remark 7.)  |
| FTEMP<br>(3,5)   | Finish temperature, use with IFINISH. (Real; enter only if ienhan = 1, Default = 0.0 or Remark 7.)  |
| SFACT<br>(3,6)   | Scale factor for time step changes other than changes due to user criteria. (Real) (Default = 1.2 or Remark 7.; enter only if ienhan = 1)   |
| IFLAG<br>(3,7)   | Enter flag to override CREEP and DYNAMIC parameters as specified in the Marc input parameter section for this load case. (Integer)<br>0 Do not override parameters.<br>1 Turn off CREEP and DYNAMICS.<br>2 Turn off CREEP.<br>3 Turn off DYNAMICS.  |
| IDTAB<br>(3,8)   | Table ID scaling damping factor (see next item) (Integer)   |

| Descriptor        | Meaning   |
|-------------------|---|
| DAMP<br>(3,9)     | Damping factor for artificial damping. The number entered here depends on the IDAMP option.<br>If IDAMP is 1, the damping matrix is scaled by setting this factor to be the ratio of the initial damping energy to the initial strain energy (Defaults to 1e-5).<br>If IDAMP is 2, the damping matrix is directly scaled by this factor.<br>If IDAMP is 4, the estimated total damping energy in the subcase will be this factor times the estimated total strain energy. (Default value of 2.e-4 or Remark 7.) |
| IDMPFLG<br>(3,10) | Enter 1 to put states reached by the above IDMPFLG flag on the post file. (Integer)   |
| IPHYS<br>(3,12)   | Flag to determine if automatic physical criteria should be added and how analysis should proceed if they are not satisfied. (Integer)<br>2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied.<br>1 Add automatic physical criteria. Stop when any user criteria are not satisfied.<br>-1 Add automatic physical criteria. Continue when any user criteria are not satisfied.<br>-2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied.   |
| I313<br>(3,13)    | Flag to check if dynamic integration error checks should be made while determining the timestep (single step Humbolt and Newmark-Beta only) (Integer, Default = 0 or Remark 7.).<br>0 Skip error check.<br>1 Include error check.   |
| CRITERIA<br>(4,1) | Enter an integer corresponding to the criteria desired. See Remark 1. (Integer; no Default)   |
| SETID<br>(4,2)    | Case Control Set ID of nodes or elements for which this criteria will apply. Restriction: Must be one of first 25 sets entered in the case control. Leave blank if "ALL" is desired. (Integer; Default is all or Remark 7.)   |
| Y1<br>(4,3)       | Time step adjustment value. See Remark 10. (Real)   |
| X1<br>(4,4)       | Time step adjustment value. See Remark 10. (Real)   |
| Y2<br>(4,5)       | Time step adjustment value. See Remark 10. (Real)   |
| X2<br>(4,6)       | Time step adjustment value. See Remark 10. (Real)   |
| Y3<br>(4,7)       | Time step adjustment value. See Remark 10. (Real)   |
| X3<br>(4,8)       | Time step adjustment value. See Remark 10. (Real)   |

| Descriptor   | Meaning   |
|--------------|---|
| Y4<br>(4,9)  | Time step adjustment value. See Remark 10. (Real) |
| X4<br>(4,10) | Time step adjustment value. See Remark 10. (Real) |

**Remarks:**

- Enter the following index in the CRITERIA field (a limit of 9 criteria may be specified and the usual option is to specify none).

|          |                        |    |                                   |
|----------|------------------------|----|-----------------------------------|
| 1        | Strain Increment       | 2  | Plastic Strain Increment          |
| 3        | Creep Strain Increment | 4  | Normalized Creep Strain Increment |
| 5        | Stress Increment       | 7  | Strain Energy Increment           |
| 8        | Temperature Increment  | 9  | Displacement Increment            |
| 10       | Rotation Increment     | 12 | Stress                            |
| 13*100+n | State Variable n       |    |                                   |

- Values entered on NLAUTO override values with the same meaning if entered elsewhere (for example, on the NLPARM or entry).
- If the NLAUTO entry is used, there should also be a corresponding NLPARM or . The matching NLPARM entry must have KMETHOD=AUTO or ITER. If is the matching entry, then field 6 must be blank or have the value ADAPT.
- Values such as (3,7) indicated corresponding item on Marc's AUTOSTEP data block 3 field 7.
- Items (3,7) to (3,13) were implemented starting with MSC Nastran 2004.0.4 and are not in previous versions.
- The continuation lines may be omitted if not needed. If one of the continuation lines is needed, all proceeding continuation lines must be entered and at least one value per line is specified (no blank lines are allowed).
- If an NLAUTO field is blank for the second and following subcases, the value will be assumed to be the same as that of the proceeding subcase for the same field. If this is not the behavior that is desired, be sure not to leave fields blank that should vary between the current and previous subcases.
- Dynamics TINIT and TFINAL are determined from and TABLED1 entries. They are ignored if entered using NLAUTO. For statics, TINIT and TFINAL, if specified on NLAUTO override the values from NLPARAM.
- Fixed time stepping is triggered by setting NSMAX equal to the number of steps (N) and IENHAN=2. It may also be triggered by using bulk data PARAM,MARCITER,N.
- The time step is adjusted based upon:

```

if   X<X1      Y(calculated)/Y1
if   X1<X<X2    Y(calculated)/Y2
if   X2<X<X3    Y(calculated)/Y3
if   X3<X<X4    Y(calculated)/Y4

```

where

| Criterion | X                | Y                                     |
|-----------|------------------|---------------------------------------|
| 1         | Strain           | Strain increment                      |
| 2         | Plastic strain   | Plastic strain increment              |
| 3         | Creep strain     | Creep strain increment                |
| 4         | Creep strain     | Creep strain increment/Elastic strain |
| 5         | Stress           | Stress increment                      |
| 7         | Strain energy    | Strain energy increment               |
| 8         | Temperature      | Temperature increment                 |
| 9         | Displacement     | Displacement increment                |
| 10        | Rotation         | Rotation increment                    |
| 12        | Stress           | Stress increment/Stress               |
| 13*100+n  | State variable n | Increment in state variable n         |

**NLBSH3D**

## Nonlinear 3-dimensional rotor loading

Defines connectivity of a three-dimensional nonlinear load as a function of relative deflection and velocity of a grid pair which in turn is a function of rotor speed.

Format:

| 1       | 2   | 3  | 4  | 5     | 6     | 7     | 8 | 9 | 10 |
|---------|-----|----|----|-------|-------|-------|---|---|----|
| NLBSH3D | SID | GA | GB | Blank | LID-1 | LID-2 |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| SID        | Nonlinear load set identification number. (Integer > 0; Required) |
| GA         | Inner grid. (Integer > 0; Required)                               |
| GB         | Outer grid. (Integer > 0; Required)                               |
| LID-1      | Load vs deflection group ID of a LBSH3DG (Integer > 0; Required)  |
| LID-2      | Load vs velocity group ID of a LBSH3DG (Integer > 0; Required)    |

**Remarks:**

1. NLBSH3D are selected with the Case Control command NONLINEAR = SID.
2. In order to ensure correct results, GRID points GA and GB must satisfy the following three conditions:
  - a. They must both be grid points
  - b. They must be coincident
  - c. They must have parallel displacement coordinate systems
 The program checks for the above conditions. If they are not satisfied, the program will terminate the run with an appropriate fatal message.
3. Multiple NLBSH3D may reference the same SID but require different GA, GB grid pairs.
4. Multiple NLBSH3D entries may refer to the same LID independent of SID.
5. Non-linear load is calculated as a function of relative deflection and velocity between GRID points GA and GB and as function of rotor speed. Current non-linear load = load (deflection (rotor speed)). Used with LBSH3DG entries to define a nonlinear load.
6. GA and GB must have all 6 DOF in D-set. Both grids should not be dependent on any RBE or MPC and must not be constrained by an SPC. The program checks for active DOFs of both the grids. If they are not satisfied, the program will terminate the run with an appropriate fatal message.

**NLCYSYM**

Defines Information to Perform Nonlinear Cyclic Symmetry Analysis using SOL 600

A limited capability is available to analyze structures with a geometry and a loading varying periodically about a symmetry axis. This capability is termed Nonlinear Cyclic Symmetry. This is done by developing a special set of tying constraints on the boundaries of the “pie-sliced” segment being analyzed. For more details on the type of structures that may be analyzed using this method and its restrictions, please see the Remarks.

**Format:**

| 1       | 2   | 3   | 4   | 5       | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-----|-----|---------|---|---|---|---|----|
| NLCYSYM | ID  | DC1 | DC2 | DC3     | X | Y |   |   |    |
|         | ANG | TOL | IRB | LinFlag |   |   |   |   |    |

**Examples:**

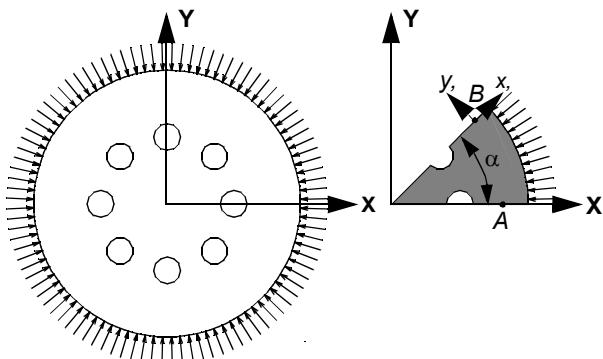
|         |      |     |     |     |     |     |  |  |  |
|---------|------|-----|-----|-----|-----|-----|--|--|--|
| NLCYSYM | 0.0  | 0.0 | 1.0 | 0.0 | 0.0 | 0.0 |  |  |  |
|         | 30.0 |     | -1  | -1  |     |     |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number not presently used. (Integer or blank, Default = 0)   |
| DC1        | First component of the direction cosine of the symmetry axis. (Real; Default = 0.0)   |
| DC2        | Second component of the direction cosine of the symmetry axis. (Real; Default = 0.0)  |
| DC3        | Third component of the direction cosine of the symmetry axis. (Real; Default = 0.0)   |
| X          | X coordinate of a point on the symmetry axis. (Real; Default = 0.0)   |
| Y          | Y coordinate of a point on the symmetry axis. (Real; Default = 0.0)   |
| Z          | Z coordinate of a point on the symmetry axis. (Real; Default = 0.0)   |
| ANG        | Cyclic angle in degrees. (Real; no Default)   |
| TOL        | Cyclic symmetry tolerance. (Real; Default = 0.5 times the minimum element size)   |
| IRB        | Rigid body mode suppression flag. (Integer; Default = -1) <ul style="list-style-type: none"> <li>-1      Automatically suppress rigid body mode.</li> <li>0      Do not suppress the rigid body mode.</li> <li>N      Suppress rigid body mode at grid point N. (N &gt; 0)</li> </ul> |
| LinFlag    | Linearization flag to be used if a cyclic symmetric structure consists of quadratic elements. (Integer; Default = 1)  |

| Descriptor | Meaning  |
|------------|--|
| 1          | The outer boundary of the structure is described based on the corner nodes only. Multipoint constraints due to cyclic symmetry are not assigned to midside nodes. Instead, they are linearly tied to the corresponding corner nodes. |
| -1         | The outer boundary of the structure is described using a quadratic field. Due to cyclic symmetry, full quadratic multipoint constraints are set up; they are assigned both to corner and midside nodes.                              |

**Remarks:**

1. A full and “pie-shaped” structure to be analyzed using nonlinear cyclic symmetry shown below:



Looking at points and on this segment, the displacement vectors should fulfill:

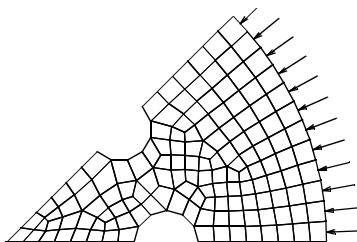
$$u'_B = u_A$$

which can also be written as:

$$u_B = R u_A$$

where the transformation matrix  $R$  depends on the symmetry axis (which, in the example above, coincides with the global Z-axis) and the sector angle. The input for the nonlinear cyclic symmetry option consists of the direction vector of the symmetry axis, a point on the symmetry axis and the sector angle. The following items should be noted:

- a. The meshes do not need to line up on both sides of a sector (for example, see the following figure)



- b. Any shape of the sector sides is allowed provided that upon rotating the sector  $360/\alpha$  times about the symmetry axis over the sector angle will result in the complete model.
- c. The nonlinear cyclic symmetry option can be combined with standard contact. In this case, both sides of the cyclic symmetry sectors need to belong to the same contact body.
- d. The nonlinear cyclic symmetry option can be combined with global remeshing (not currently available).
- e. In a coupled thermo-mechanical analysis, the temperature is forced to be cyclic symmetric (TA=TB) (not currently available).
- f. A nodal point on the symmetry axis is automatically constrained in the plane perpendicular to the symmetry axis.
- g. Possible rigid body motion about the symmetry axis can be automatically suppressed.
- h. Cyclic Symmetry is valid for:
  - Primarily for continuum elements. However, the presence of beams and shells is allowed, but there is not connection of shells to shells across the symmetry plane, so the shell part can, for example, be a turbine blade and the volume part is a turbine rotor. The blade is connected to the rotor and if there are 20 blades, 1/20 of the rotor is modeled plus one complete blade.
  - Nonlinear static analysis including remeshing as well as coupled analysis
  - Valid for all analysis involving contact
  - Valid also for: eigenvalue analysis such as buckling or modal extraction, and linear transient dynamic analysis.
  - Cyclic Symmetry is invalid for pure heat transfer.
  - The contact status can be viewed. Grids tied across the symmetry plane will have a value of 2.

To prevent confusion, it must be emphasized that the cyclic symmetry feature described above is different than linear cyclic symmetry commonly used in modal analysis where physical quantities such as  $x_n$ , displacements, forces, stresses and temperature in the n-th segment are expanded in a Fourier series with terms of the cyclic components,  $u^k$ , in the fundamental region, like:

$$x_n = \frac{1}{N} u^0 + \sqrt{\frac{2}{N}} \sum_{k=1}^{K} [u^{k,c} \cos(n-1)k\alpha + u^{k,s} \sin(n-1)k\alpha] + \frac{(-1)^{n-1}}{\sqrt{N}} u^{N/2}$$

where  $k$  is the harmonic order;  $N$  is the total number of sectors;  $\alpha$  is the fundamental inter-sector phase shift defined as  $2\pi/N$ ; and  $K$  is defined as:

$$K = \begin{cases} \frac{N-1}{2} & \text{if } N \text{ is odd} \\ \frac{N-2}{2} & \text{if } N \text{ is even} \end{cases}$$

There are considerable savings in both computing time and data storage associated with the use of the linear cyclic symmetry concept. Assuming a finite element model with a sector size of  $m$  degrees of freedom, a real-valued cyclic symmetry approach leads, in the worst case, to one eigenvalue problem of size  $m$  and  $(N-1)/2$  eigenvalue problems of size  $2m$ . A complex approach leads to  $N$  eigenvalue problems of size  $m$ ; while the full analysis leads to a single, but very costly, eigenvalue problem of size  $Nm$ .

Although linear cyclic symmetry can reduce the problem size greatly, it is restricted to linear analysis, and the sector must have its surface mesh on the symmetry planes to be identical on each side of the sector. The nonlinear cyclic symmetry implemented in SOL 600 can be used for nonlinear problems, such as contact, and the nodes do not need to line up on both symmetry planes of the sector.

**NLDAMP****Damping Constants - SOL 600**

Defines damping constants for nonlinear analysis when Marc is executed from SOL 600 only.

**Format:**

| 1      | 2    | 3    | 4     | 5    | 6     | 7 | 8 | 9 | 10 |
|--------|------|------|-------|------|-------|---|---|---|----|
| NLDAMP | EID1 | EID2 | ALPHA | BETA | GAMMA |   |   |   |    |

**Examples:**

|        |   |      |      |     |     |  |  |  |  |
|--------|---|------|------|-----|-----|--|--|--|--|
| NLDAMP | 1 | 2000 | .025 | 4.5 | 1.0 |  |  |  |  |
|--------|---|------|------|-----|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID1       | First element for which the damping values will be used. (Integer $\geq 0$ ; Required) |
| EID2       | Last element for which the damping values will be used. (Integer $\geq 0$ or blank)    |
| ALPHA      | Mass Matrix Multiplier. (Real; Default = 0.0)  |
| BETA       | Stiffness Matrix Multiplier. (Real; Default = 0.0)                                     |
| GAMMA      | Numerical Damping Multiplier. (Real; Default = 0.0)                                    |

**Remarks:**

1. This entry matches Marc's Damping definition.
2. NLDAMP is recognized only when Marc is executed from SOL 600.

**NLFREQ**

## Frequency List for Nonlinear Harmonic Response

Defines explicit forcing frequencies for nonlinear harmonic response.

Format:

| 1      | 2  | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|----|----|--------|----|----|----|----|----|----|
| NLFREQ | ID | F1 | F2     | F3 | F4 | F5 | F6 | F7 |    |
|        | F8 | F9 | -etc.- |    |    |    |    |    |    |

Examples:

|        |    |    |    |     |     |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|
| NLFREQ | 17 | 3. | 7. | 21. | 14. |  |  |  |  |
|--------|----|----|----|-----|-----|--|--|--|--|

**Descriptor**      **Meaning**

ID      Identification number referenced by the NLFREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0)

F1...Fn      Forcing frequency values in cycles per unit time. (Real  $\geq 0.0$ )

Remarks:

1. The frequencies may be specified in any order; they do not have to be in ascending or descending order.
2. Only one NLFREQ or NLFREQ1 entry with the same ID is allowed.

**NLFREQ1**

## Frequency List for Nonlinear Harmonic Response, Alternate Form 1

Defines a set of forcing frequencies for nonlinear harmonic response by specification of a starting frequency, frequency increment, and the number of increments/decrements desired.

**Format:**

| 1       | 2  | 3  | 4  | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|----|----|----|-----|---|---|---|---|----|
| NLFREQ1 | ID | F1 | DF | NDF |   |   |   |   |    |

**Examples:**

|         |    |    |    |   |  |  |  |  |  |
|---------|----|----|----|---|--|--|--|--|--|
| NLFREQ1 | 17 | 3. | 2. | 6 |  |  |  |  |  |
|---------|----|----|----|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number referenced by the NLREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0). |
| F1         | First forcing frequency in the set. (Real $\geq 0.0$ )   |
| DF         | Frequency increment. See Remark 1. (Real $<> 0.0$ ; Required)  |
| NDF        | Number of frequency increments/decrements. (Integer > 0; Default = 1)                                      |

**Remarks:**

1. If DF is positive, it defines a frequency increment. In this case, the first excitation frequency will be F1 with all subsequent excitations frequencies being of larger value than F1. If DF is negative, it defines a frequency decrement. In this case, the first excitation frequency will still be F1, but now all subsequent excitations frequencies will be of smaller value than F1. In both cases, the initial conditions for a frequency are the response configuration at the previous excitation frequency. The initial conditions for the first frequency are zero.
2. Only one NLREQ or NLFREQ1 entry with the same ID is allowed.

**NLHARM**

## Parameters for Nonlinear Harmonic Response Control

Defines parameters for nonlinear harmonic response analysis.

**Format:**

| 1      | 2  | 3      | 4     | 5      | 6 | 7 | 8 | 9 | 10 |
|--------|----|--------|-------|--------|---|---|---|---|----|
| NLHARM | ID | SUBFAC | NHARM | NLFREQ |   |   |   |   |    |

**Examples:**

|        |    |  |   |    |  |  |  |  |  |
|--------|----|--|---|----|--|--|--|--|--|
| NLHARM | 41 |  | 3 | 15 |  |  |  |  |  |
|--------|----|--|---|----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number referenced by the NLHARM Case Control command. (Integer > 0)                        |
| SUBFAC     | Factor for capturing sub-harmonic response. See Remark 3. (Integer $\geq 1$ ; Default = 1)                |
| NHARM      | The number of harmonics to include in the solution. See Remark 2. (Integer > 0)                           |
| NLFREQ     | Identification number of the NLFREQ or NLFREQ1 entry specifying the forcing frequency list. (Integer > 0) |

**Remarks:**

1. The NLHARM ID must be unique among all NLHARM entries.
2. The greater the degree of nonlinearity, the larger the number of harmonics required to find a solution (if one exists). The number of harmonics (NHARM) used may influence the ability of the algorithm to find solutions to the nonlinear problem posed. However, the larger the number of harmonics, the more computational effort is required to find a solution.
3. The response frequencies in linear harmonic analysis are the same as the forcing frequency. Permanent oscillations whose frequencies are a fraction of the forcing frequency ( $\frac{1}{2}, \frac{1}{4}, \dots$ ) may occur in a nonlinear system; these oscillations are known as sub-harmonic response.

**NLHEATC**

Defines Numerical Analysis Parameters for SOL 600 Heat Transfer Analysis

Format:

| 1       | 2    | 3     | 4    | 5    | 6    | 7     | 8 | 9 | 10 |
|---------|------|-------|------|------|------|-------|---|---|----|
| NLHEATC | ID   | MLS   | MRC  | MMIN | NPOS | IASMB |   |   |    |
|         | TCHG | TEVAL | TERR |      |      |       |   |   |    |

Examples:

|         |      |         |     |   |   |  |  |  |  |
|---------|------|---------|-----|---|---|--|--|--|--|
| NLHEATC | 9999 | 10      | 1   | 1 | 1 |  |  |  |  |
|         | 10.  | 999999. | 20. |   |   |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Not presently used, leave blank.  |
| MLS        | Maximum number of load steps in the run. (Integer; Default = 4). [2,1]  |
| MRC        | Maximum number of recycles during an increment due to temperature dependent material properties. (Integer; Default = 3) [2,2]   |
| MMIN       | Minimum number of recycles during an increment. This value is forced even if convergence appears to occur in a fewer number of cycles. (Integer; Default = 1) [2,3]   |
| NPOS       | Nonpositive definite flag. (Integer; Default = 0) [2,7] <ul style="list-style-type: none"> <li>0 Solution of nonpositive definite matrices will fail.</li> <li>1 Solution of nonpositive definite matrices is forced (computer time is greater if nonpositive matrices do not exist)</li> </ul> |
| IASMB      | Flag to assemble conductivity matrix. (Integer; Default = 0) [2,12] <ul style="list-style-type: none"> <li>0 Conductivity matrix is not assembled each iteration</li> <li>1 Conductivity matrix is assembled each iteration</li> </ul>  |
| TCHG       | Maximum nodal temperature change allowed. (Real; Default = 20.0) [3,1]  |
| TEVAL      | Maximum temperature change before properties are re-evaluated and matrices reassembled. (Real; Default = 100.0) [3,2]   |
| TERR       | Maximum error in temperature estimate used for property evaluation. It provides a recycling capability to improve accuracy for highly nonlinear heat transfer such as latent heat and radiation. (Real; Default = 0.0) [3,3]  |

Remarks:

- Only one NLHEATC entry should be entered.
- This entry maps to Marc's CONTROL history definition entry for heat transfer. [i,j] indicates the datablock and field of this Marc entry.

**NLMOPTS****Nonlinear Multiple Options**

Specifies nonlinear material options and composite options. The nonlinear material options are for advanced materials used in SOL400. The composite options (TSHEAR and INLAM, CPROJ) are applicable to all solution sequences in which Layered Solid Elements (PCOMPLS) are available, namely, SOL600, SOL400, SOL200 (analysis only), and all linear solution sequences between SOL101 and SOL112.

**Format:**

| 1       | 2          | 3       | 4      | 5      | 6       | 7        | 8     | 9      | 10 |
|---------|------------|---------|--------|--------|---------|----------|-------|--------|----|
| NLMOPTS | "CREEP"    | valc1   | valc2  | valc3  | valc4   |          |       |        |    |
|         | "ASSM"     | vala    |        |        |         |          |       |        |    |
|         | "TSHEAR"   | vals    |        |        |         |          |       |        |    |
|         | "LRGSTRN"  | valle   |        |        |         |          |       |        |    |
|         | "HEMICUBE" | Value   | NPIXEL |        | CUTOFF  | FRACTION | FACCN | FACTOL |    |
|         | "TEMPP"    | valtd   |        |        |         |          |       |        |    |
|         | "TEMGO"    | vmaptg  |        |        |         |          |       |        |    |
|         | "SPROPMAP" | PROPMAP | PROPBH | DIRECT | THICKOP | IPRINT   |       |        |    |
|         | "SPCRMPT"  | vramp   |        |        |         |          |       |        |    |
|         | "DEACTEL"  | vald1   | vald2  |        |         |          |       |        |    |
|         | "ENTHALP"  | valclu  | valen1 |        |         |          |       |        |    |
|         | "MAPTOL"   | vmptol  |        |        |         |          |       |        |    |
|         | "INLAM"    | vcoord  | cproj  |        |         |          |       |        |    |

**Examples:**

|         |          |   |     |  |   |      |  |  |  |
|---------|----------|---|-----|--|---|------|--|--|--|
| NLMOPTS | CREEP    | 0 |     |  |   |      |  |  |  |
|         | HEMICUBE | 1 | 500 |  | 0 | 0.01 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| “CREEP”    | Keyword indicating that the formulation for creep analysis. (Character Default CREEP). Creep data should be entered using MATVP, not the CREEP bulk data entry.<br><br>There are 2 kinds of creep models<br><br>(a) Explicit Creep - ValC1 and ValC2 are used for this.  |
| valc1      | 0 : Maxwell Model Creep (default)<br>1 : Explicit Viscoplastic Creep (only supported via user subroutine CRPLAW)   |
| valc2      | (Blank or 0) : (Default)<br>1: (Reserved)  |
|            | (b) Implicit Creep - ValC3 and ValC4 are used for this.  |
| valc3      | 0 : Implicit Creep off<br>1 : Implicit Creep on  |
| valc4      | (Blank or 0) : elastic tangent (Default)<br>1: secant tangent<br>2: radial return  |
|            | Note that:<br>(1) ValC3 = 0 means Explicit Creep (see ValC1). ValC3 = 1 means Implicit Creep.<br>(2) ValC4 is only used in conjunction with Valc3=1. The recommended value for ValC4 is 1 (secant tangent).  |
| “ASSM”     | Keyword indicating that the item following applies to assumed strain. See Remark 3. (Character; Default = See Remark 3.)   |
| vala       | ASSUMED for assumed strain formulation<br>OFF for no assumed strain formulation.   |
| “TSHEAR”   | Keyword indicating that the item following applies to a parabolic shear distribution through the shell thickness (or ply thickness). See Remark 4.   |
| vals       | TSHEAR for parabolic distribution. (Character; Default = blank)  |
| “LRGSTRN”  | Keyword indicating that the item following applies to a formulation for large strain. (Default; Integer = 0)<br><br>-1 No large strain formulation.<br>0 Mean normal return.<br>1 Hypoelasticity and additive plasticity with mean normal return.<br>2 Hyperelasticity and multiplicative plasticity with radial return. |
| “HEMICUBE” | Keyword to select the view factor calculation method. See Remarks 2. and 7.  |

| Descriptor | Meaning  |   |
|------------|--|---|
| Field 3    | VALUE=0 - Use Nastran finite difference, contour integration, or Gaussian integration method (Default).  | =1 - (HEMI) Use pixel based modified hemicube method. This method is not available for axisymmetric view factor calculations. |
| Field 4    | NPIXEL - Enter the number of pixels (Default = 500).   |   |
| Field 6    | CUTOFF - Enter the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored (Default = 0).  |   |
| Field 7    | FRACTION - Enter the fraction of the maximum view factors that is to be treated implicitly. View factors values smaller than this cutoff are treated explicitly (Default = 1.0E-2).  |   |
| Field 8    | FACCNT - Set 1 to activate explicit treatment of reflection matrix. Default = 0)   |   |
| Field 9    | FACTOL - Tolerance to be used on the above iteration on the Poljak equations.  |   |
| “TEMPP”    | Keyword indicating temperature distribution across the thickness of advanced heat shell elements. Internal invisible scalar points will be created to store these extra variables. LINE and QUAD options require shell elements to have full 3-D thermal material data. When the temperature results of these internal points are applied to calculating thermal loading effect in a separate mechanical analysis, this keyword must also be specified in the structural model (See Remarks 11., 12. and 13.). |   |
| valtd      | CONS   | Constant distribution. (Default)  |
|            | LINE   | Linear distribution. (Character)  |
|            | QUAD   | Quadratic distribution.   |
| “TEMGO”    | Keyword indicating user request identified in .f06 file, the mapping of user supplied and internally generated grids for the linear and quadratic thermal shell elements. See Remark 8.  |   |
| vmaptg     | NO   | Do not show grid mapping. (Character; Default)  |
|            | YES  | Show mapping.   |
| “SPROPMAP” | Keyword controlling automated inclusion of nonlinear property options for SOL 400. See Remark 9.   |   |
| PROPMAP    | (Integer $\geq 0$ ; Default 0)   |   |
|            | -1   | No mapping considered   |
|            | 0  | Map to full integration elements associated with nonlinear property extension entries when necessary                          |
|            | 1  | Map to reduced integration elements associated with nonlinear property extension entries when necessary                       |

| Descriptor | Meaning  |
|------------|--|
|            | <p>2 Map to full integration elements associated with nonlinear property extension entries when possible</p> <p>3 Map to reduced integration elements associated with nonlinear property extension entries when possible</p> <p><b>Note:</b> The nonlinear property extension entries refer to: PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 entries.</p> <p>To ensure that the automatic mapping is unique for an element ID, whenever advanced elements are detected / specified by the user, ELTPRT will be automatically called.</p> |
| PROPBEH    | Property behavior flag. Valid values are 1D, 3D, PLST, PSTR, AXI. (Default = None for 1D, 3D. PLST for 2D; Character)  |
| DIRECT     | Layer direction flag. Valid values; 1, 2, 3. (Integer > 0; Default 1)  |
| THICKOP    | For 2D solids: out of plane thickness. (Real $\geq 0.0$ ; Default 1.0)   |
| IPRINT     | Print flag for generated property flags.   |
|            | <p>0 Blank (Default)</p> <p>1 Print</p> <p>2 Punch</p> <p>3 Print and Punch</p>  |
| “SPCRMPT”  | Keyword indicating heat transfer SPC is to be applied instantaneously or by linear step. (Character Default SPCMPT)  |
| vramp      | <p>0 SPC applied instantaneously. (Integer or blank; Default = blank)</p> <p>1 SPC applied by linear step.</p>   |
| “DEACTEL”  | Keyword controlling default behavior and output diagnostics for element deactivation   |
| vald1      | Select default deactivation rules for each physics pass in the analysis. See Remark 9.   |
|            | <p>-1 Default deactivation rules are always ignored. (Default in single physics analysis)</p> <p>0 Default deactivation rules are always applied. (Default in coupled analysis)</p> <p>1 Default deactivation rules are applied in a mechanical pass only.</p> <p>2 Default deactivation rules are applied in a thermal pass only.</p>   |
| vald2      | Select diagnostic output behavior for active and deactivated elements.   |
|            | <p>0 A summary of all deactivated elements is printed. If no elements get deactivated output is skipped. (Default)</p> <p>1 A detailed list of all deactivated elements is printed. If no elements get deactivated output is skipped.</p> <p>2 A summary of all deactivated and all remaining active elements is printed.</p>  |

| Descriptor | Meaning   |
|------------|---|
| 3          | A detailed list of all deactivated and all remaining active elements is printed.  |
| "ENTHALP"  | Keyword controlling the scheme used to form the capacitance matrix and the enthalpy vector for elements with nonlinear property extensions in transient heat transfer. See Remark 10. |
| valclu     | Select option to form capacitance matrix for elements with nonlinear property extensions.   |
| 0          | Use same scheme as that for linear elements without property extensions   |
| 1          | Use consistent capacitance matrix   |
| 2          | Use coupled capacitance matrix  |
| 3          | Use lumped capacitance matrix   |
| valen1     | Select option to form enthalpy vector for elements with nonlinear property extensions   |
| 0          | Form enthalpy using automatic scheme. (Default)   |
| 1          | Form enthalpy explicitly through time integration for material behavior   |
| 2          | Form enthalpy via multiplication of temperature with capacitance  |
| "MAPTOL"   | Keyword controlling the tolerance used to check if a structural node lies inside a thermal element.   |
| vmptol     | Value of MAPTOL. (Real > 0.0; Default = 0.2)  |
| INLAM      | (Character; Default Blank)  |
| vcoord     | Selects coordinate system for interlaminar stress output.   |
| Blank      | Using the original coordinate system for interlaminar stress output   |
| BOTT       | Using the ply coordinate system of bottom layer of the interlaminates   |
| TOP        | Using the ply coordinate system of above layer of the interlaminates  |
| cproj      | Activates projection system for projecting X-axis of CORDM onto layer plane. See remark 5.  |
| OFF        | Does not project (Default).   |
| ON         | Projection scheme is activated.   |

**Remarks:**

1. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.

2. The material and property related keywords associated with the entry (CREEP, ASSM, TSHEAR, LRGSTRN, TEMPP and ENTHALP) are only applicable to elements associated with PAXISYM, PBARN1, PBEMN1, PRODN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMPLS, and PCOHE entries. Other keywords SPCRMPT, HEMICUBE, DEACTEL, MAPTOL) are more generally applicable. Keyword SPCRMPT applies to all SPC temperature boundary conditions, HEMICUBE applies to radiation boundary conditions and DEACTEL applies to all elements on the job.
3. The setting of ASSUMED STRAIN in SOL 400 is automatic for the following three elements with some conditions.
  - a. 4 nodes full integration Plane Stress (PLPLANE + PSHLN2 with PSTRS -- BEH4=PSTRS, INT4=L);
  - b. 4 nodes full integration Plane Strain (PLPLANE + PSHLN2 with PSTRN -- BEH4=PLSTRN, INT4=L); and
  - c. 8 nodes full integration HEXA (PSOLID + PSLDN1 -- BEH8=SOLID, INT8=L,)

The application conditions are summarized here.

**OFF**              Assumed strain always turned off.

**ASSUMED**      Assumed strain will be turned on manually for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:

- a. Elastic / Elasto-Plastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1). This includes isotropic and non-isotropic materials.
- b. Elastic / Elasto-Plastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1). This includes isotropic and non-isotropic materials.

Assumed strain flag will be turned off for all materials using the Multiplicative Formulation (NLMOPTS,LRGSTRN,2).

**Default**          Assumed Strain will be turned on automatically for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:

- a. Isotropic Elastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1)
- b. Isotropic Elastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1)

Note that this turns on Total Lagrange Formulation currently by default for the isotropic elastic materials.

Assumed Strain will be turned off automatically for the following cases:

- a. Any material that uses Updated Lagrange in either Additive or Multiplicative Formulation
- b. Any non-isotropic material

4. TSHEAR allows a parabolic shear distribution for the BEH4=DCT, INT4=L or LRIH (shell) elements, and BEH8 or BEH20=SLCOMP, INTi=L, Q (Layered Solid) or ASTN (Layered Solid Shell) elements.
5. In Figure 9-114, plane ABCD is the layer plane of interest.  $Z_m$  is normal to the layer plane (or is the thickness direction).  $X_m$  is the projection of the X axis of the MCID coordinate system on the layer plane.  $Z_m \times X_m$  gives the  $Y_m$ . Angle  $\theta$  is measured counter-clockwise from  $X_m$  axis about the  $Z_m$  axis.

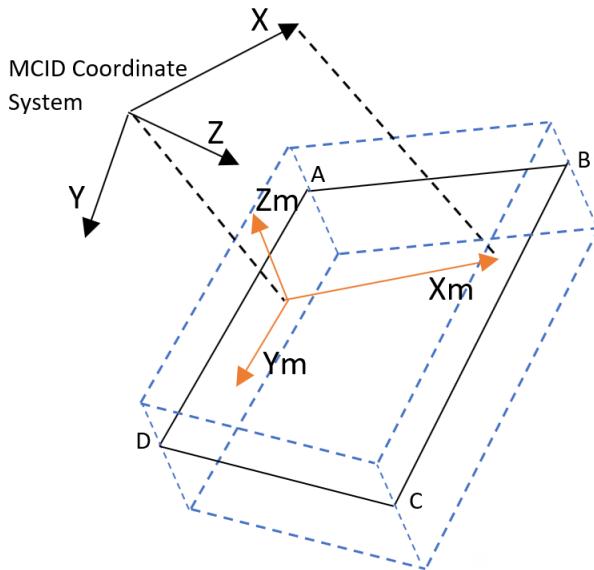


Figure 9-114      Projection Scheme

6. NLMOPTS,LRGs,valu can be used separately or in conjunction with the PARAM,LGDISP option. Two separate cases can be identified:
  - a. Only NLMOPTS,LRGs specified. In this case, valle = -1 and valle = 0 mean the same, i.e., no large strain flags are turned on for the applicable elements identified in remark 2. Valle = 1 flags the additive framework and valle = 2 flags the multiplicative framework.
  - b. Both NLMOPTS,LRGs and PARAM,LGDISP,N ( $N > -1$ ) specified. In this case, valle = -1 supercedes the PARAM,LGDISP option and turns off all large displacement / large strain flags for the applicable elements identified in remark 2. Also, valle = 0 behaves the same way as valle = 1 (i.e., for the applicable elements, both options flag the additive framework).
7. Definition of the radiation exchange matrix as in the *MSC Nastran Thermal User's Guide*, Eq. 0-15.

$$R = \sigma [A\epsilon - A\alpha(A - F(I - \alpha))^{-1}F\epsilon]$$

in which the reflection matrix is:

$$[A - F(I - \alpha)]^{-1}$$

Note that the reflection term is costly, since it involves the factorization of a dense matrix.

For the SOL 400 Newton's method, the previous exchange matrix multiplied by a function of the temperature is added to the stiffness. If the exchange matrix is dense, which is generally the case, the sparse stiffness matrix consequently also becomes dense, and the factorization of the stiffness matrix becomes much more expensive.

The input options are as follows. All options with the exception of the "faccnt" option, are available in SOLs 153, 159, and 400. The "faccnt" option is only available in SOL 400:

hemi, view, npixel, ndiv, cutoff, fraction, faccnt, factol

where

|          |  |
|----------|--|
| view     | 1 to flag hemicube method. Default is 0. The following options are available only if view = 1.   |
| npixel   | Number of pixels per quarter section, where total number of pixels = $(2 \cdot \text{npixel})^2$ . Default is 500.   |
| ndiv     | Not used, but originally intended for axisymmetry. Reserve this one for the future to do adaptive mesh refinement, with ndiv defined as the maximum allowable number of subdivisions.  |
| cutoff   | Factor below which viewfactors will be set to zero. This one is applied during the last viewfactor calculation. It results in a less dense exchange matrix and therefore a faster calculation of the reflection term and of the stiffness matrix factorization. Default is 0.0.  |
| fraction | Factor below which radiation exchange matrix terms are not added to the stiffness matrix. For each equation in the radiation exchange matrix, the fraction is multiplied by the diagonal term, and all terms in the equation less than this value is not added to the stiffness matrix. This procedure does not affect the density of the radiation exchange matrix or the cost of calculating it, including the reflection term, but reduces the density of the stiffness matrix and the cost of factorizing the stiffness matrix. Note that the full radiation exchange matrix still is used to calculate fluxes. Default is 0.01. |
| faccnt   | Set to 1 to activate explicit treatment of reflection matrix. Default is 0. In this procedure, the radiation exchange matrix is constructed to be:<br>$R = \sigma[A\varepsilon - A\alpha F\varepsilon]$  |

Note that the expensive reflection term is absent. This reduced form of the radiation exchange matrix is added to the stiffness matrix. Since the reflection matrix never is calculated and factorized, the calculation of the radiation exchange matrix is significantly cheaper. Since this reduced radiation exchange matrix is less dense, the factorization of the stiffness matrix also is significantly faster.

For the flux calculation, an iterative procedure is used based on iterating towards a solution simultaneously satisfying both Poljak equations in the *MSC Nastran Thermal User's Guide*, Eqs. 6-11 and 6-12, respectively.

$$A\{q\}_e^{IN} = [F]\{q\}_e^{OUT}$$

$$\{q\}_e^{OUT} = \sigma[\varepsilon]\{u_e\}^4 + [I - \varepsilon]\{q\}_e^{IN}$$

The value of “faccnt” also can be set to the maximum allowable number of iterations to be used in this procedure. If the value is set to 1, the maximum allowable number is internally set to 100. This procedure currently is available only in SOL 400 and is not available for wavelength-dependent emissivities.

The “cutoff” and “fraction” parameters can be specified concurrently with this option.

factol tolerance to be used on the previous iterative procedure on the Poljak equations.

#### 8. TEMGO,YES results in the following type of print.

| USER GRID POINT(TOP) MAPPING TO BOT/MID INTERNAL GRID |           |           |           |           |           |           |           |           |           |           |
|---|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|
| TOP   | 1         | 2         | 3         | 4         | 5         | 6         | 7         | 8         | 9         | 10        |
| BOT   | 101000001 | 101000002 | 101000003 | 101000004 | 101000005 | 101000006 | 101000007 | 101000008 | 101000009 | 101000010 |
| MID   | 0         | 0         | 0         | 0         | 0         | 0         | 0         | 0         | 0         | 0         |
| ...   |           |           |           |           |           |           |           |           |           |           |

9. The “SPROPMAP” keyword provides a convenient option to automatically flag secondary properties like PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2, and PSLDN1. Note that these secondary property entries expose the user to a set of sophisticated 2-D continuum and 3-D beam, shell and continuum elements in SOL 400. The rules governing the generation of the additional properties are many and are a function of the problem dimension, material type, and procedure. All these rules have been incorporated into the automatic generation option. These are briefly summarized in the table below for the default SPROPMAP = 0 case:

| Secondary Property | Primary Property | Dimension | Material | Notes  | Unsupported Features of Primary Entry |
|--------------------|------------------|-----------|----------|--------|---------------------------------------|
| PBARN1             | PBARL            | 1-D       | MAT4     | Note 4 |                                       |
|                    |                  |           | MATS1    | Note 1 |                                       |
|                    |                  |           | MATEP    |        |                                       |
|                    |                  |           | MATF     |        |                                       |
|                    |                  |           | MATSMA   | Note 2 |                                       |
|                    |                  |           | MATVE    |        |                                       |
|                    |                  |           | MATVP    | Note 3 |                                       |

| Secondary Property | Primary Property | Dimension | Material | Notes  | Unsupported Features of Primary Entry |
|--------------------|------------------|-----------|----------|--------|---------------------------------------|
| PBEMN1             | PBEAML           | 1-D       | MAT4     | Note 4 | Tapered Sections                      |
|                    |                  |           | MATS1    | Note 1 |                                       |
|                    |                  |           | MATEP    |        |                                       |
|                    |                  |           | MATF     |        |                                       |
|                    |                  |           | MATSMA   | Note 2 |                                       |
|                    |                  |           | MATVE    |        |                                       |
|                    |                  |           | MATVP    | Note 3 |                                       |
| PRODN1             | PROD             | 1-D       | MAT4     | Note 4 | J,C                                   |
|                    |                  |           | MATS1    | Note 1 |                                       |
|                    |                  |           | MATEP    |        |                                       |
|                    |                  |           | MATF     |        |                                       |
|                    |                  |           | MATSMA   | Note 2 |                                       |
|                    |                  |           | MATVE    |        |                                       |
|                    |                  |           | MATVP    | Note 3 |                                       |
| PSHEARN            | PSHEAR           | 3-D       | MAT4     | Note 4 | F1,F2                                 |
|                    |                  |           | MAT8     |        |                                       |
|                    |                  |           | MATS1    | Note 1 |                                       |
|                    |                  |           | MATS8    | Note 1 |                                       |
|                    |                  |           | MATEP    |        |                                       |
|                    |                  |           | MATF     |        |                                       |
|                    |                  |           | MATORT   |        |                                       |
|                    |                  |           | MATSMA   | Note 2 |                                       |
|                    |                  |           | MATVE    |        |                                       |
|                    |                  |           | MATVP    | Note 3 |                                       |
| PSHLN1             | PSHELL           | 3-D       | MAT4     | Note 4 | TS/T, nondefault Z1 and Z2, $121/T^3$ |
|                    |                  |           | MAT5     | Note 4 |                                       |
|                    |                  |           | MATS1    | Note 1 |                                       |
|                    |                  |           | MATS8    | Note 1 |                                       |
|                    |                  |           | MATEP    |        |                                       |
|                    |                  |           | MATF     |        |                                       |
|                    |                  |           | MATORT   |        |                                       |
|                    |                  |           | MATSMA   | Note 2 |                                       |
|                    |                  |           | MATVE    |        |                                       |
|                    |                  |           | MATVP    | Note 3 |                                       |

| Secondary Property | Primary Property | Dimension | Material | Notes  | Unsupported Features of Primary Entry               |
|--------------------|------------------|-----------|----------|--------|---|
| PSHLN1             | PCOMP/<br>PCOMPG | 3-D       | MAT4     | Note 4 | FT, GE, LAM options other than BLANK and SYM, SOUTi |
|                    |                  |           | MAT5     | Note 4 |   |
|                    |                  |           | MAT8     |        |   |
|                    |                  |           | MATS1    | Note 1 |   |
|                    |                  |           | MATS2    | Note 1 |   |
|                    |                  |           | MATS8    | Note 1 |   |
|                    |                  |           | MATEP    |        |   |
|                    |                  |           | MATF     |        |   |
|                    |                  |           | MATORT   |        |   |
|                    |                  |           | MATSMA   | Note 2 |   |
|                    |                  |           | MATVE    |        |   |
|                    |                  |           | MATVP    | Note 3 |   |
| PSHLN2             | PLPLANE          | 2-D       | MAT4     | Note 4 |   |
|                    |                  |           | MAT5     | Note 4 |   |
|                    |                  |           | MATG     | Note 5 |   |
|                    |                  |           | MATS1    | Note 1 |   |
|                    |                  |           | MATS3    | Note 1 |   |
|                    |                  |           | MATS8    | Note 1 |   |
|                    |                  |           | MATEP    |        |   |
|                    |                  |           | MATF     |        |   |
|                    |                  |           | MATORT   |        |   |
|                    |                  |           | MATSMA   | Note 6 |   |
|                    |                  |           | MATVE    |        |   |
|                    |                  |           | MATVP    | Note 7 |   |
| PSLDN1             | PSOLID           | 3-D       | MAT4     | Note 4 | IN, ISOP, FCTN                                      |
|                    |                  |           | MAT5     | Note 4 |   |
|                    |                  |           | MATS1    | Note 1 |   |
|                    |                  |           | MATEP    |        |   |
|                    |                  |           | MATF     |        |   |
|                    |                  |           | MATORT   |        |   |
|                    |                  |           | MATSMA   |        |   |
|                    |                  |           | MATVE    |        |   |
|                    |                  |           | MATVP    |        |   |

|       |  |
|-------|--|
| Note: | 1. Only when NLMOPTS, LRGSTRN, 1 is flagged or HGENPLAS is provided NLSTEP entry for coupled analysis.             |
|       | 2. For shape memory materials, define through MATSMA only the thermo-mechanical model is available.                |
|       | 3. For creep, define through MATVP, VALC=0 must be set on NLMOPTS for explicit formulation.                        |
|       | 4. Only when phase changes are flagged.  |
|       | 5. Gasket materials for BEH4=COMPS or AXCOMP, INT4=L.  |
|       | 6. For shape memory materials, define through MATSMA and BEH=PLSTRS only the thermo-mechanical model is available. |
|       | 7. For creep, define through MATVP and BEH=PLSTRS VALC=0 must be set on NLMOPTS for explicit formulation.          |
|       | 8. The “SPROPMAP” does not support MATDIGI.  |

In the previous table, the Secondary Property (Column 1) is automatically flagged depending on the Primary Property (Column 2) and the Primary/Secondary Materials (Column 4). The Comments column (Column 5) indicates any special conditions that are followed by the program while doing this mapping. The Unsupported Features column (Column 6) indicates the list of options on the Primary Property entry that are not supported by the Secondary Property.

It should be noted that the Secondary Property additions offer a wide range of element formulations that primarily cater to nonlinear extension but can also serve for linear applications. By default, the PROPMAP field is set to 0 and maps to the full integration elements of the secondary property only if necessary. For a less memory intensive element, the PROPMAP field can be manually set to 1 and this then maps to the equivalent reduced integration element of the secondary property only if available and necessary. PROPMAP field can be manually set to 2 or 3 when secondary properties need to be tagged to all elements in the model. In this case, the special conditions given in the 5th column above are ignored and the mapping is carried out for both linear and nonlinear materials (if allowed). PROPMAP = 2 maps to full integration elements whenever possible and PROPMAP = 3 maps to reduced integration elements whenever possible.

- a. Property mapping is considered for all possible elements (i.e., PROPMAP = 0/1 is automatically converted to PROPMAP = 2/3) for the following conditions:
  - When IDAMP (damping scheme specified through the NLSTEP entry) > 0 is used for static analysis. Note that the damping scheme is only available for elements with nonlinear property extensions.
  - When CRITTID (user criteria specified through the NLSTEP entry) and the associated TABSCTL entry refer to displacement / stress / strain criteria for all elements in the model. Note that the user criteria are only supported for elements with nonlinear property extensions.

- When a small displacement analysis is flagged for thermo-mechanical coupled analysis specified through SUBSTEP or for real-time thermo-mechanical chained analysis specified through TEMP(LOAD,HSUB,HSTEP,HTIME). Note that for real-time temperatures to be transferred from the heat pass to the structural pass, all elements need to be processed in the nonlinear program flow and this is accomplished by flagging all possible elements with nonlinear property extensions.
- b. Property mapping is provided for elements that are associated with special procedures / flags:
  - Elements whose grids are specified on the VCCT Bulk Data entry. VCCT is only supported for grids of elements with nonlinear property extensions.
  - Elements that are specified on the IPSTRN / ISTRESS commands. These are only supported for elements with nonlinear property extensions.

It should be noted also that while the NLMOPTS,SPROPMAP option is powerful and allows a user-friendly way to map to advanced elements, it does not provide a substitute for all cases. Defaults are provided for the most common cases on the NLMOPTS,SPROPMAP entry itself. This includes PROPBEN - especially useful to distinguish between plane strain and plane stress, DIRECT - the layer thickness direction for gaskets, THICKOP - the out-of-plane thickness for the planar case. It should be noted that these defaults are global defaults and apply to all the elements for which the secondary properties are added. If such defaults are not applicable (for e.g., different out-of-plane thicknesses), it is the user's responsibility to add individual secondary property entries.

IPRINT can be used to get additional information on the secondary properties that have been added. Default is 0 in which case, there is no print-out. IPRINT = 1 allows the print-out in the .f06 file, IPRINT = 2 allows the print-out in the .pch file, IPRINT = 3 allows print-out in both the .f06 and .pch files.

| Example 1: 8 noded CHEXA, PSOLID, MAT1 |  |
|--|--|
| SPROPMAP                               | Added Secondary Property   |
| -1                                     | No mapping considered  |
| 0                                      | No mapping done - not necessary                                    |
| 1                                      | No mapping done - not necessary                                    |
| 2                                      | PSLDN1: C8, BEH8 = SOLID, INT8 = L, BEH8H = SOLID, INT8H = L       |
| 3                                      | PSLDN1: C8, BEH8 = SOLID, INT8 = LRIH, BEH8H = SOLID, INT8H = LRIH |

### Example 2: 4 noded CQUAD4, PSHELL, MATF

| SPROPMAP | Added Secondary Property                                    |
|----------|---|
| -1       | No mapping (note that MATF will not work in this case)      |
| 0        | PSHLN1: C4, BEH4 = DCT, INT4 = L, BEH4H = DCT, INT4H = L    |
| 1        | PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L |
| 2        | PSHLN1: C8, BEH8 = DCT, INT4 = L, BEH4H = DCT, INT4H = L    |
| 3        | PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L |

9. The **vald1** value in “DEACTEL” determines in which physics pass the default deactivation rules are applied. If **vald1** is positive the deactivation rules are applied in the specified pass only and in all other passes the defaults are switched off, meaning that in those passes all elements remain active unless a DEACTEL Bulk Data input deactivates some of them.

The default deactivation rules are summarized in the following table:

| Element Types  | By Default Deactivated in Pass |
|--|--------------------------------|
| Connector elements CWELD, CFAST, CSEAM                             | Thermal                        |
| Rigid body elements Rxxx   | Thermal                        |
| CONMi and CMASSi elements  | Thermal                        |
| Interface elements CIFxxx  | Thermal                        |
| Surface elements CHBDYx  | Mechanical                     |
| Heat convection elements CONVx                                     | Mechanical                     |
| Radiation boundary elements RADBC                                  | Mechanical                     |
| CDAMP4 elements originating from CMB or CMS fields in BCBODY input | Mechanical                     |

The default for elements of all other types is that they are active in every pass.

10. The “ENTHALP” keyword is applicable to elements associated with the nonlinear property extensions PRODN1, PSHEARN, PSHLN1, PSLDN1 and for axisymmetric heat transfer shells specified through PAXISYM. It provides different options to form the capacitance matrix and enthalpy vector for these elements through **valclu** and **valenl** respectively.
- The **valclu** value allows the choice of 3 different capacitance formulations: consistent, coupled (average of consistent and lumped) and lumped. The default value of **valclu** = 0 allows the nonlinear elements to follow the same scheme as that for linear elements; lumped for rods and 3D shells, coupled for axisymmetric shells and linear hexes, pentas, tetras and consistent for quadratic hexes, pentas and tetras. Values of **valclu** > 0 allow the user to pick specific capacitance formulations for all elements.

The **valenl** value allows the user the choice of 2 different enthalpy vector formulations: time integration of the enthalpy at each integration point or product of the nodal temperature with the capacitance. The default value of **valenl = 0** allows the program to pick automatically: use the time integration option for nonlinear specific heat / latent heat, use the nodal product option for linear specific heat. Values of **valenl > 0** allow the user to pick the approach. Note that for nonlinear problems, the time integration approach is more accurate.

11. To compute the thermal loads of 3D thermal shells in a separate mechanical run, a consistent temperature distribution type must be specified in mechanical models, i.e. LINE must be defined if the thermal model has a linear temperature distribution using LINE, or QUAD must be specified for the thermal model with a quadratic temperature distribution defined by QUAD. If LINE or QUAD is specified, the program will switch the structural shell elements to advanced elements automatically by generating PSHLN1 entries internally if no PSHLN1 or PSHLN2 entries are specified in the original model.
12. MSC NASTRAN thermal analysis does not differentiate between top and bottom faces while applying loads or boundary conditions on shell, even though the top and bottom temperatures may be different through specifying TEMPP = LINE or QUAD.
13. TEMPP keyword is supported in coupled analysis or in separate runs of thermal and mechanical analyses. This keyword is not supported in chained thermal and mechanical analyses.

**NLOUT**

Selects Additional Nonlinear Output Quantities as Referenced By NLSTRESS Case Control Command

Selects additional nonlinear output quantities as referenced by NLSTRESS Case Control Command in SOL 400.

**Format:**

| 1     | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|------|------|------|------|------|------|------|----|
| NLOUT | ID   | AIO1 | AIO2 | AIO3 | AIO4 | AIO5 | AIO6 | AIO7 |    |
|       | AIO8 | AIO9 | etc. |      |      |      |      |      |    |
|       | ESV  | SV2  | SV3  | etc. |      |      |      |      |    |

**Example:**

|       |   |         |  |  |  |  |  |  |  |
|-------|---|---------|--|--|--|--|--|--|--|
| NLOUT | 1 | TOTTEMP |  |  |  |  |  |  |  |
|-------|---|---------|--|--|--|--|--|--|--|

|       |     |         |     |  |  |  |  |  |  |
|-------|-----|---------|-----|--|--|--|--|--|--|
| NLOUT | 1   | TOTTEMP |     |  |  |  |  |  |  |
|       | ESV | SV2     | SV3 |  |  |  |  |  |  |

**Descriptor**      **Meaning**

- ID              Identification number of a NLOUT entry. Selected by the NLOUT keyword on the NLSTRESS Case Control command. (Integer > 0)
- AIOi            Additional Output request. See Remark 3. (Character)
- ESV            Keyword indicating the following are user state variables, it is optional.
- SVi            State variable names selected for output.

**Remarks:**

1. The keywords can appear in any order.
2. Any item selected that is inappropriate for the element will be ignored.
3. The request codes, meaningful only to elements who refer to PRODN1, PBARN1, PBEMN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMLS, or PCOHE entries, are as follows:

| Additional Output Code Keywords |                              |
|---------------------------------|------------------------------|
| Keyword                         | Description                  |
| CCASTRSS                        | Components of Cauchy Stress  |
| CTOTSTRN                        | Components of Total Strain   |
| CELASTRN                        | Components of Elastic Strain |
| CPLASTRN                        | Components of Plastic Strain |

| Additional Output Code Keywords |  |
|---------------------------------|--|
| Keyword                         | Description  |
| CCRPSTRN                        | Components of Creep Strain   |
| CTHMSTRN                        | Components of Thermal Strain   |
| TSTRNPS                         | Thickness Strain for Plane Stress  |
| MAJESTRN                        | Major Engineering Strain   |
| MINESTRN                        | Minor Engineering Strain   |
| CURVOL                          | Current Volume   |
| ORGVOL                          | Original Volume  |
| TOTTEMP                         | Total Temperature  |
| INCTEMP                         | Incremental Temperature  |
| EQVMSTRS                        | Equivalent von Mises Stress  |
| EQSTRSA                         | Equivalent Stress/Yield Stress Ratio                                     |
| EQELSTRN                        | Equivalent Elastic Strain  |
| EQPLSTRN                        | Equivalent Plastic Strain  |
| EQCRSTRN                        | Equivalent Creep Strain  |
| TTSTRNED                        | Total Strain Energy Density  |
| ELSTRNED                        | Elastic Strain Energy Density  |
| PLSTRNED                        | Plastic Strain Energy Density  |
| PLSTRNRT                        | Plastic Strain Rate  |
| ILNMSTRS                        | Interlaminar Normal Stress. See Remark 5.                                |
| ILSHSTRS                        | Interlaminar Shear Stress. See Remark 5.                                 |
| ILSHTKCX                        | X-component of the interlaminar shear stress for thick composite shells. |
| ILSHTKCY                        | Y-component of the interlaminar shear stress for thick composite shells  |
| CSTRSCRD                        | Components of Stress Preferred System                                    |
| GSKTCLST                        | Gasket Pressure  |
| GSKTCLSR                        | Gasket Closure   |
| PGSKTCLS                        | Plastic Gasket Closure   |
| FAILIDX                         | Failure Index (%)  |
| TOTVSV1                         | Total Value of First State Variable                                      |
| TOTVSV2                         | Total Value of Second State Variable                                     |
| TOTVSV3                         | Total Value of Third State Variable                                      |

| Additional Output Code Keywords |  |
|---------------------------------|--|
| Keyword                         | Description                            |
| EQPHSTRN                        | Equivalent phase transformation strain |
| EQTWSTRN                        | Equivalent TWIN strain                 |
| EQTPSTRN                        | Equivalent TRIP strain 75              |
| CPHSTRN                         | Phase transformation strain tensor     |
| VOLFMART                        | Volume fraction of Martensite          |

4. User state variable name following the ESV keyword can be the default nominal name or user defined name in UDSESV.
5. ILNMSTRS and ILSHSTRS are stress tensors formed in the basic coordinate system, their principal directions are the directions of stress vector of interlaminar normal and shear stresses respectively. More precisely, the tensor components  $\sigma_{ij}$  satisfy below equation:

$$(\sigma_{ij} - \delta_{ij}\sigma)n_j = 0$$

where;

$n_j$  is the stress vector direction,

$\sigma$  is the normal of stress vector, and

$\delta_{ij} = 0$  when  $i \neq j$  and  $\delta_{ij} = 1$  when  $i = j$ .

**NLOUTUD****User Defined Output Requests For Nonlinear Explicit Analysis.**

User defined output requests for elements or Lagrangian grid points. Use in SOL700 only.

**Format:**

| 1       | 2      | 3      | 4       | 5     | 6     | 7       | 8 | 9 | 10 |
|---------|--------|--------|---------|-------|-------|---------|---|---|----|
| NLOUTUD | NID    | GROUP  | UNAME   | DTOUT | STEPS | OUTNAME |   |   |    |
| +       | EL1/G1 | EL2/G2 | Eli/Gli |       |       |         |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myuds 'SCA.MDSolver.Obj.Uds.Dytran.InitOut'
```

*In Bulk Data:*

|         |   |       |        |      |  |          |  |  |  |
|---------|---|-------|--------|------|--|----------|--|--|--|
| NLOUTUD | 4 | MYUDS | EEXOUT | 0.01 |  | OUTPUTR1 |  |  |  |
| +       | 1 | 2     | 3      | 4    |  |          |  |  |  |

| Field   | Contents   |
|---------|--|
| NID     | Unique output number. (Integer > 0; Required)  |
| GROUP   | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)   |
| UNAME   | Name User subroutine name associated with the entry. (Character; default=EEXOUT)<br><br>EEXOUT: Output for elements<br><br>GEXOUT: Output for Lagrangian grid points |
| DTOUT   | Output time interval. (Real > 0.0; default=blank)  |
| STEPS   | Output step interval. (Integer > 0; default=blank)   |
| OUTNAME | Name for output request. (Character; required.)  |
| Eli     | Element id for user output. (Integer > 0; required)  |
| Gi      | Grid point id for user output. (Integer > 0; required)   |

**Remarks:**

1. Only can be used for SOL 700.
2. UNAME can be EEXOUT or GEXOUT.
3. Only one of DTOUT and STEPS is available.

4. When UNAME is set to EEXOUT, element id's are required for user output and when UNAME is set to GEXOUT, grid point id's are required for user output.

5. The file names that will be generated are:

{jobname}\_{OUTNAME}\_{cycle number}.[THS,ARC]

**NLTERM**

## Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy. NLSTEP is the preferred option for SOL 400.

Format:

| 1      | 2      | 3    | 4    | 5       | 6     | 7       | 8       | 9       | 10 |
|--------|--------|------|------|---------|-------|---------|---------|---------|----|
| NLTERM | ID     | NINC | DT   | KMETHOD | KSTEP | MAXITER | CONV    | INTOUT  |    |
|        | EPSU   | EPSP | EPSW | MAXDIV  | MAXQN | MAXLS   | FSTRESS | LSTOL   |    |
|        | MAXBIS |      |      |         | MAXR  |         | RTOLB   | MINITER |    |

Example:

|        |    |   |  |      |  |  |  |  |  |
|--------|----|---|--|------|--|--|--|--|--|
| NLTERM | 15 | 5 |  | ITER |  |  |  |  |  |
|--------|----|---|--|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number. (Integer > 0)   |
| NINC       | Number of increments. See Remark 16. (Integer > 0)   |
| DT         | Incremental time interval for creep analysis. See Remark 3. (Real $\geq 0.0$ ; Default = 0.0 for no creep.)  |
| KMETHOD    | Method for controlling stiffness updates. See Remark 4. (Character = "AUTO", "ITER", "SEMI", "FNT", or "PFNT"; Default = "AUTO" for SOL 106, "AUTO" for SOL 400 with non-contact analysis, and "FNT" for SOL 400 with contact analysis.) |
| KSTEP      | Number of iterations before the stiffness update for ITER method. See Remarks 5. For the FNT and PFNT usage of KSTEP, please see Remark 19. (Integer $\geq -1$ ; Default = 5 for SOL 106 and 10 for SOL 400)                             |
| MAXITER    | Limit on number of iterations for each load increment. See Remark 6. (Integer $\neq 0$ ; Default = 25)   |
| CONV       | Flags to select convergence criteria. See Remarks 7., 21., and 22. (Character = "U", "P", "W", "V", "N", "A" or any combination; Default = "PW". (See Remark 4 for additional default comment.)  |
| INTOUT     | Intermediate output flag. See Remark 8. (Character = "YES", "NO", "ALL" or Integer > 0 for SOL 400 only; Default = NO)   |
| EPSU       | Error tolerance for displacement (U) criterion. See Remarks 4., 16., 17. and 20. (Real $> 0.0$ ; Default = 1.0E-2)   |
| EPSP       | Error tolerance for load (P) criterion. See Remarks 4., 16. and 17. (Real $> 0.0$ ; Default = 1.0E-2)  |
| EPSW       | Error tolerance for work (W) criterion. See Remarks 4., 16., 17. and 20. (Real $> 0.0$ ; Default = 1.0E-2)   |

| Descriptor | Meaning  |
|------------|--|
| MAXDIV     | Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 9. (Integer $\neq 0$ ; Default = 3)  |
| MAXQN      | Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 10. (Integer $\geq 0$ ; Default = MAXITER for all methods except PFNT. For PFNT, Default = 0)  |
| MAXLS      | Maximum number of line searches allowed for each iteration. See Remark 11. (Integer $\geq 0$ ; Default = 4 for all methods except PFNT. For PFNT, Default = 0)   |
| FSTRESS    | Fraction of effective stress ( $\bar{\sigma}$ ) used to limit the subincrement size in the material routines. See Remark 12. (0.0 < Real < 1.0; Default = 0.2)   |
| LSTOL      | Line search tolerance. See Remark 12. (0.01 < Real < 0.9; Default = 0.5)   |
| MAXBIS     | Maximum number of bisections allowed for each load increment. See Remark 13. (-10 $\leq$ MAXBIS $\leq$ 10; Default = 5 except for MAXITER < 0; Default = 0 if MAXITER < 0)   |
| MAXR       | Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark 14. (1.0 $\leq$ MAXR $\leq$ 40.0; Default = 20.0)  |
| RTOLB      | Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15. (Real > 2.0; Default = 20.0)  |
| MINITER    | Minimum number of iterations for each increment, SOL 101 with contact and SOL 400 only. (Integer $> 0$ ; Default = 1; In contact analysis, Default = 2) When high accuracy is required, it is also recommended to set MINITER = 2. |

#### Remarks:

1. The NLPARM entry is selected by the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.
2. In cases of static analysis (DT = 0.0) using Newton methods, NINC is the number of equal subdivisions of the load change defined for the subcase. Applied loads, gravity loads, temperature sets, enforced displacements, etc., define the new loading conditions. The differences from the previous case are divided by NINC to define the incremental values. In cases of static analysis (DT = 0.0) using arc-length methods, NINC is used to determine the initial arc-length for the subcase, and the number of load subdivisions will not be equal to NINC. In cases of creep analysis (DT > 0.0), NINC is the number of time step increments.
3. For creep analysis, the unit of DT must be consistent with the unit used on the CREEP entry that defines the creep characteristics. Total creep time for the subcase or step is DT multiplied by the value in the field NINC; i.e., DT\*NINC. For SOL 400 with advanced nonlinear elements (elements with PSLDN1, PSHLN1, etc.) for creep analysis, if the total time does not equal 1.0, then the NLSTEP Bulk Data entry must be used. For SOL 600 DT is ignored except for creep analyses.

4. The stiffness update strategy is selected in the KMETHOD field.
  - If the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 9. and 13. for diverging solutions.
  - If the SEMI option is selected, the program for each load increment (i) performs a single iteration based upon the new load, (ii) updates the stiffness matrix, and (iii) resumes the normal AUTO option.
  - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if  $KSTEP \leq MAXITER$ . However, if  $KSTEP > MAXITER$ , stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and  $KSTEP = MAXITER$ . The "U" convergence test is not used if ITER-1 is selected.
  - If the FNT option is selected, the program will use the full Newton iteration method for which the stiffness matrix will be updated at every iteration. FNT option is available for SOL 400 only. For SOL 106, please use "KMETHOD=ITER and KSTEP=1" instead. In comparison with the PFNT method, the defaults for FNT are EPSU=0.01, EPSW=0.01 and MAXLS=4. See Remark 19.
  - If the PFNT option is selected, the program will use the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are internally set as EPSU=-0.01, EPSW=-0.01, and MAXLS=0. The PFNT method is available for SOL 400 only. See Remark 19.
5. For AUTO and SEMI options, the stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.
6. The number of iterations for a load increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or  $MAXBIS = 0$ ) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. One best solution is computed for SOL 106 and 4 best solutions are computed for SOL 400. The analysis is terminated if the solution still diverges. If MAXDIV is negative, the analysis is terminated immediately.

For SOL 400 only, MAXITER can be negative. If MAXITER is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each load increment. The default for MAXBIS = 0, if  $MAXITER < 0$ . Also, for SOL 400, the value of MAXITER for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.
7. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W,

then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if CONV = ‘blank’ the code will use a default of “UPW” if no contact analysis and “PV” if a contact analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 21. For A, see Remark 22.

8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES or ALL must be specified in order to be able to perform a subsequent restart from the middle of a subcase.

| INTOUT | Output Processed                                      |
|--------|---|
| YES    | For every computed load increment.                    |
| NO     | For the last load of the subcase or step.             |
| ALL    | For every computed and user-specified load increment. |

- For the Newton family of iteration methods (i.e., when no NLPCI command is specified), the option ALL is equivalent to option YES since the computed load increment is always equal to the user-specified load increment.
  - For SOL 400 only, if the adaptive time stepping scheme is used (i.e., when a NLAUTO Bulk Data entry with the same ID is specified), INTOUT is allowed to be Integer > 0. In this case, the load step is divided into INTOUT increments for output. For example, if INTOUT=5, the output will be at load increments 0.2, 0.4, 0.6, 0.8, and 1.0. Please note that INTOUT defines only the output load increments, which are different from the analysis increments. The analysis load increment size is smaller than or equal to the output load increment size.
  - For arc-length methods (i.e., when the NLPCI command is specified) the computed load increment in general is not going to be equal to the user-specified load increment, and is not known in advance. The option ALL allows the user to obtain solutions at the desired intermediate load increments.
  - For SOL 600 only, the default is YES (see Remark 18.)
9. The ratio of energy errors before and after the iteration is defined as divergence rate ( $E^i$ ), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If  $E^i \geq 1$  or  $E^i < -10^{12}$ , then  $NDIV = NDIV + 2$

If  $-10^{12} < E^i < -1$ , then  $NDIV = NDIV + 1$

The solution is assumed to diverge when  $NDIV \geq |\text{MAXDIV}|$ . If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIV is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated on the second divergence.

10. The BFGS update is performed if  $\text{MAXQN} > 0$ . As many as MAXQN quasi-Newton vectors can be accumulated. The BFGS update with these QN vectors provides a secant modulus in the search direction. If MAXQN is reached, no additional ON vectors will be accumulated. Accumulated QN vectors are purged when the stiffness is updated and the accumulation is resumed.
11. The line search is performed as required, if  $\text{MAXLS} > 0$ . In the line search, the displacement increment is scaled to minimize the energy error. The line search is not performed if the absolute value of the relative energy error is less than the value specified in LSTOL.
12. The number of subincrements in the material routines (elastoplastic and creep) is determined so that the subincrement size is approximately  $\text{FSTRESS} \cdot \bar{\sigma}$  (equivalent stress). FSTRESS is also used to establish a tolerance for error correction in the elastoplastic material; i.e.,  

$$\text{error in yield function} < \text{FSTRESS} \cdot \bar{\sigma}$$

If the limit is exceeded at the converging state, the program will exit with a fatal message. Otherwise, the stress state is adjusted to the current yield surface.
13. The number of bisections for a load increment/arc-length is limited to the absolute value of MAXBIS. Different actions are taken when the solution diverges depending on the sign of MAXBIS. If MAXBIS is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If MAXBIS is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after  $|\text{MAXBIS}|$  bisections, the analysis is continued or terminated depending on the sign of MAXDIV. See Remark 9.
14. MAXR is used in the adaptive load increment/arc-length method to define the overall upper and lower bounds on the load increment/arc-length in the subcase; i.e.,

$$\frac{1}{\text{MAXR}} \leq \frac{\Delta l_n}{\Delta l_o} \leq \text{MAXR}$$

where  $\Delta l_n$  is the arc-length at step  $n$  and  $\Delta l_o$  is the original arc-length. The arc-length method for load increments is selected by an NLPCI Bulk Data entry. This entry must have the same ID as the NLPARM Bulk Data entry.

15. The bisection is activated if the incremental rotation for any degree-of-freedom ( $\Delta\theta_x$ ,  $\Delta\theta_y$ , or  $\Delta\theta_z$ ) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.
16. The default for NINC is 10, except if there is a GAP, Line Contact, Heat Transfer or PARAM,NLTOL,0, in which case the default is 1. Default tolerance sets are determined based on model type and desired accuracy. Accuracy is under user control and can be specified on the PARAM, NLTOL entry. For SOL 106, NLTOL's value is used only if the CONV, EPSU, EPSP and EPSW

fields are blank, and if NINC is set to a value of 10 or larger. Otherwise, the NLTERM selection will be overridden. The overridden values are CONV=PW, EPSP=1.0E-3, and EPSW=1.0E-7. For SOL 400, NLTERM's value is used if the value of CONV, EPSU, EPSP, or EPSW fields are blank. The tables below list tolerances according to NLTERM selections:

**Table 25 Default Tolerances for Static Nonlinear SOL 106 Models Without Gaps, Contact or Heat Transfer**

| NLTOL | Designation   | CONV | EPSU  | EPSP   | EPSW   |
|-------|---------------|------|-------|--------|--------|
| 0     | Very high     | PW   | _____ | 1.0E-3 | 1.0E-7 |
| 1     | High          | PW   | _____ | 1.0E-2 | 1.0E-3 |
| 2     | Engineering   | PW   | _____ | 1.0E-2 | 1.0E-2 |
| 3     | Prelim Design | PW   | _____ | 1.0E-1 | 1.0E-1 |
| None  | Engineering   | PW   | _____ | 1.0E-2 | 1.0E-2 |

**Table 26 Default Tolerances for Static Nonlinear SOL 106 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)**

| NLTOL | Designation | CONV | EPSU  | EPSP   | EPSW   |
|-------|-------------|------|-------|--------|--------|
| 0     | Very high   | PW   | _____ | 1.0E-3 | 1.0E-7 |
| 2     | Engineering | PW   | _____ | 1.0E-3 | 1.0E-5 |
| None  | Engineering | PW   | _____ | 1.0E-3 | 1.0E-5 |

**Table 27 Default Tolerances for Static Nonlinear SOL 106 or 153 Models With Heat Transfer (Enter NLTOL Value of 0 Only or Omit the Parameter)**

| NLTOL | Designation | CONV | EPSU  | EPSP   | EPSW   |
|-------|-------------|------|-------|--------|--------|
| 0     | Very high   | PW   | _____ | 1.0E-3 | 1.0E-7 |
| None  | Very high   | PW   | _____ | 1.0E-3 | 1.0E-7 |

**Table 28 Default Tolerances for Static Nonlinear SOL 400 Models Without Gaps, Contact or Heat Transfer**

| NLTOL | Designation   | CONV | EPSU | EPSP   | EPSW   |
|-------|---------------|------|------|--------|--------|
| 0     | Very high     | PW   | -    | 1.0E-3 | 1.0E-3 |
| 1     | High          | PW   | -    | 1.0E-2 | 1.0E-3 |
| 2     | Engineering   | PW   | -    | 1.0E-2 | 1.0E-2 |
| 3     | Prelim Design | PW   | -    | 1.0E-1 | 1.0E-1 |
| None  | Engineering   | PW   | -    | 1.0E-2 | 1.0E-2 |

Table 29 Default Tolerances for Static Nonlinear SOL 400 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

| NLTOL | Designation | CONV | EPSU | EPSP   | EPSW   |
|-------|-------------|------|------|--------|--------|
| 0     | Very high   | PW   | -    | 1.0E-3 | 1.0E-3 |
| 2     | Engineering | PW   | -    | 1.0E-3 | 1.0E-3 |
| None  | Engineering | PW   | -    | 1.0E-3 | 1.0E-3 |

17. The method to compute the energy (work) error is different for SOL 106 and SOL 400. For SOL 106, the energy error is computed based on the residue forces. While, for SOL 400, the energy error computed is the total energy error, which is based on the nonlinear forces acting on the structure. At the start of the iteration, these two methods give approximately the same value. However, near convergence, the SOL 106 method will field a much smaller value than that provided by the SOL 400 method. The difference in these two methods is reflected in the default values shown in Remark 16. The reason for a new method used in SOL 400 is that it gives the true error of the physical energy. On the other hand, the error computed in SOL 106 has no counter part in the physical world.
18. For SOL 600, the only fields used are ID, NINC, DT (creep only), KMETHOD and INTOUT, however, PARAM,MARCOTIM is recommended instead of INTOUT. For other fields, advanced convergence controls are available using NLAUTO, NLSTRAT and PARAM,MARCDATA Bulk Data entries. For SOL 600, if INTOUT is specified all NLPARM's in the file must use the same values. The first INTOUT encountered is what is actually used. The default for INTOUT is YES. For SOL 600, the initial time step for each subcase is 1/NINC of the NLPARM applicable to that subcase. If TINIT or the NLAUTO entry is entered it overrides 1/NINC as the initial time step. For arc length methods NLPCI with the same ID as NLPARM must be entered and if AIFRACT or the NLSTRAT entry is entered it will override 1/NINC as the initial increment size. Beware that NLSTRAT entries, if used, must be entered for each subcase as well as for "subcase zero". The ID of NLSTRAT do not correspond to the NLPARM Id or to the subcase ID but are numbered sequentially starting with zero for Marc increment zero, one for the first subcase (regardless of its ID) etc. For KMETHOD only, strings AUTO and ITER are supported. If any other string is entered it will be assumed to be the same as AUTO in SOL 600.
19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed with respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.

21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.
- By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.
22. For SOL 400, the convergence checking flag "A" is implemented. "A" means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force is small, i.e., PVA → UVA . The legal combinations for A and PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example, PVA → UVA , PVA → UNA , etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.
23. For SOL 101 contact, NLPARM can be used to control nonlinear solution process, such as the number of load increments. Since the only source of nonlinearity comes from contact, some fields, however, are neither relevant to the iterative solution nor considered as user-controllable. These fields include DT(=0.0), KMETHOD(=AUTO), KSTEP(=500), INTOUT(=NO) and MAXLS(=0). NLPARM is not required for running SOL 101 contact jobs. If it is not provided by the user, Nastran will create one. All the values of its fields are printed in F06 file under N O N - L I N E A R I T E R A T I O N S O L U T I O N C O N T R O L P A R A M E T E R S. Listed below are the remaining defaults for SOL 101 contact analysis.

| NINC | MAXITER | CONV | MAXDIV | MAXQN | FSTRESS | LSTOL | MAXBIS | MAXR | RTOLB | MINITER |
|------|---------|------|--------|-------|---------|-------|--------|------|-------|---------|
| 10   | 25      | "PV" | 3      | 0     | 0.2     | 0.5   | 5      | 20.0 | 20.0  | 2       |

Note that the defaults for EPSU, EPSP and EPSW are different for NINC < 10 and NINC ≥ 10.

|           | EPSU  | EPSP  | EPSW  |
|-----------|-------|-------|-------|
| NINC < 10 | 1.e-3 | 1.e-3 | 1.e-7 |
| NINC ≥ 10 | 1.e-2 | 1.e-2 | 1.e-2 |

24. If Modules are present then this entry may only be specified in the main Bulk Data section.

**NLPCI****Parameters for Arc-Length Methods in Nonlinear Static Analysis**

Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106 and SOL 400). This entry will be used if a subcase contains an NLPARM command (NLPARM = ID).

**Format:**

| 1     | 2  | 3    | 4      | 5      | 6     | 7 | 8       | 9     | 10 |
|-------|----|------|--------|--------|-------|---|---------|-------|----|
| NLPCI | ID | TYPE | MINALR | MAXALR | SCALE |   | DESITER | MXINC |    |

**Example:**

|       |    |      |     |     |  |  |    |    |  |
|-------|----|------|-----|-----|--|--|----|----|--|
| NLPCI | 10 | CRIS | 1.0 | 1.0 |  |  | 12 | 10 |  |
|-------|----|------|-----|-----|--|--|----|----|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number of an associated NLPARM entry. (Integer > 0)   |
| TYPE       | Constraint type. See Remark 2. (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”)   |
| MINALR     | Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (0.0 < Real $\leq$ 1.0; Default = 0.25) |
| MAXALR     | Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (Real $\geq$ 1.0; Default = 4.0)        |
| SCALE      | Scale factor (w) for controlling the loading contribution in the arc-length constraint. SOL 106 only. (Real $\geq$ 0.0; Default = 0.0)                             |
| DESITER    | Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. See Remarks 3. and 4. (Integer > 0; Default = 12)                  |
| MXINC      | Maximum number of controlled increment steps allowed within a subcase. See Remark 5. (Integer > 0; Default = 20)   |

**Remarks:**

1. The NLPCI entry is selected by the Case Control command NLPARM = ID. There must also be an NLPARM entry with the same ID. However, for creep analysis (DT  $\neq$  0.0 in NLPARM entry), the arc-length methods cannot be activated, and the NLPCI entry is ignored if specified. The NLPCI entry is not recommended for heat transfer analysis in SOL 153. Arc-length method is not supported in contact analysis.
2. The available constraint types are as follows:

TYPE = “CRIS”:

$$\{u_n^i - u_n^0\}^T \{u_n^i - u_n^0\} + w^2 (\mu^i - \mu^0)^2 = \Delta l_n^2$$

TYPE = "RIKS":

$$\{u_n^i - u_n^{i-1}\}^T \{u_n^i - u_n^0\} + w^2 \Delta \mu^i = 0$$

TYPE = "MRIKS":

$$\{u_n^i - u_n^{i-1}\}^T \{u_n^{i-1} - u_n^0\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^0) = 0$$

where:

w = the user-specified scaling factor (SCALE)

$\mu$  = the load factor

$\Delta l$  = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor  $\mu$ . As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite w, the arc-length method is degenerated to the conventional Newton's method.

3. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one load increment to the next by:

$$\text{MINALR} \leq \frac{\Delta l_{\text{new}}}{\Delta l_{\text{old}}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use MINALR = MAXALR = 1.

4. The arc-length  $\Delta l$  for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous load increment ( $I_{\max}$ ) and the number of iterations desired for convergence in the current load increment (DESITER) as follows:

$$\Delta l_{\text{new}} = \Delta l_{\text{old}} \sqrt{\frac{\text{DESITER}}{I_{\max}}}$$

5. The MXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. This field is useful in limiting the number of increments computed for a collapse analysis.
6. NLPCI does not support general contact in SOL 400.

**NLRGAP****Nonlinear Load Proportional to Gap**

Defines a nonlinear radial (circular) gap for transient response or nonlinear harmonic response.

**Format:**

| 1      | 2   | 3  | 4  | 5     | 6    | 7    | 8    | 9      | 10 |
|--------|-----|----|----|-------|------|------|------|--------|----|
| NLRGAP | SID | GA | GB | PLANE | TABK | TABG | TABU | RADIUS |    |

**Example:**

|        |    |   |   |    |   |    |   |     |  |
|--------|----|---|---|----|---|----|---|-----|--|
| NLRGAP | 21 | 3 | 4 | XY | 3 | 10 | 6 | 1.6 |  |
|--------|----|---|---|----|---|----|---|-----|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Nonlinear load set identification number. (Integer > 0)   |
| GA         | Inner (e.g., shaft) grid for radial gap. (Integer > 0)  |
| GB         | Outer (e.g., housing) grid for radial gap. (Integer > 0)  |
| PLANE      | Radial gap orientation plane: XY, YZ, or ZX. (Character; Default = XY.)   |
| TABK       | TABLED1 ID defining gap stiffness vs. time or frequency (Integer > 0) or the TABLED1 ID defining gap force vs. penetration (Integer < 0), or TABLED5 ID defining frequency and (gap force vs. penetration) for nonlinear harmonic response only. See Remark 11. |
| TABG       | Table ID for radial gap clearance as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)  |
| TABU       | Table ID for radial coefficient of friction as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)  |
| RADIUS     | Shaft radius. (Real $\geq$ 0.0; Default = 0.0)  |

**Remarks:**

1. NLRGAP must be selected with the Case Control command NONLINEAR = SID.
2. Multiple NLRGAP entries with the same SID are allowed.
3. The NLRGAP is not an element, but a nonlinear load similar to the NOLINi Bulk Data entries. It computes the relative displacements of GA and GB in the selected plane and applies appropriate nonlinear loads to simulate the radial contact.
4. The degrees-of-freedom in the XY, YZ, or ZX planes (depending on the PLANE) of GA and GB must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation. If RADIUS is > 0.0, then the in-plane rotation degree-of-freedom must also be in the solution set.

5. The NLRGAP is limited to use in direct transient or nonlinear harmonic response solution sequences. When enforced motion is used, the NLRGAP requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
6. The XY, YZ and ZX planes are relative to the displacement coordinate systems of GA and GB. In order to ensure correct results, points GA and GB must satisfy the following three conditions:
  - a. They must both be grid points
  - b. They must be coincident
  - c. They must have parallel displacement coordinate systemsThe program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message. The coincident check can be skipped by specifying system(648)=1. The default is 0.
7. The shaft radius is used only for the computation of friction induced torque.
8. In the underlying equations, a positive coefficient of friction is consistent with counter-clockwise shaft rotation from axis 1 towards axis 2 (anti-clockwise). A negative coefficient of friction is consistent with clockwise shaft rotation from axis 2 towards axis 1 (clockwise). See [Figure 9-115](#).
9. Nonlinear forces for the grids referenced on the NLRGAP can be output with the NLLOAD Case Control command. See [Figure 9-115](#) for the sign conventions.

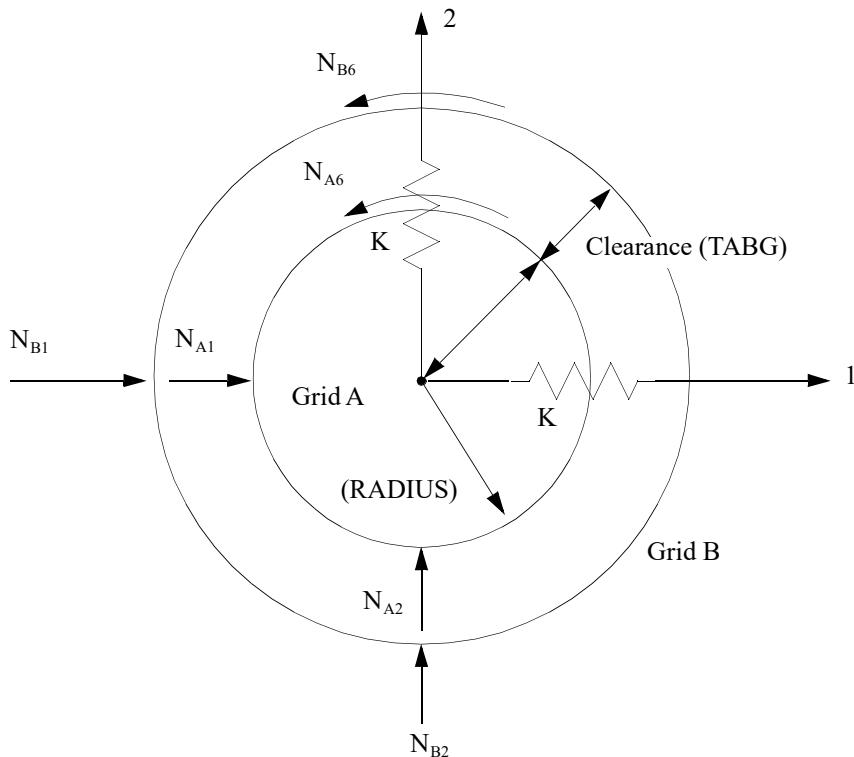


Figure 9-115 Radial Gap Orientation and Nonlinear Load Sign Conventions

10. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

11. TABK may reference either a TABLED1 ID or a TABLED5 ID.

For transient response, only a TABLED1 entry may be referenced. There are two possible scenarios:

- a. The number entered in the TABK field is a positive integer. In this case the integer references a TABLED1 entry defining pairs of time vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer. In this case the absolute value of the integer references a TABLED1 entry defining pairs of gap penetration vs. gap force.

For nonlinear harmonic response either a TABLED1 entry or a TABLED5 entry may be referenced leading to three possible scenarios:

- a. The number entered in the TABK field is a positive integer referencing a TABLED1 entry. In this case, the pairs of values defined on the TABLED1 entry are frequency vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer, the absolute value of which references a TABLED1 entry. In this case, the pairs of values defined on the TABLED entry are gap penetration vs. gap force.

- c. The number entered in the TABK field is a positive integer referencing a TABLED5 entry. In this case, the pairs of values defined on the TABLED5 entry are frequency vs. a TABLED1 ID that defines pairs of gap penetration vs. gap force.
- 12. Forces due to TABK and TABU at GA and GB are only present when the gap is closed. A moment is applied only when the gap is closed and RADIUS > 0.0.

**NLRSFD****Nonlinear Transient Load Proportional to a Squeeze Film Damper**

Defines a nonlinear transient radial squeeze film damper.

**Format:**

| 1      | 2       | 3       | 4          | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|---------|---------|------------|--------|--------|--------|--------|--------|----|
| NLRSFD | SID     | GA      | GB         | PLANE  | BDIA   | BLEN   | BCLR   | SOLN   |    |
|        | VISCO   | PVAPCO  | NPORT      | PRES1  | THETA1 | PRES2  | THETA2 | NPNT   |    |
|        | OFFSET1 | OFFSET2 | GROUP NAME | NAME2  |        |        |        |        |    |
|        | RDATA1  | RDATA2  | RDATA3     | RDATA4 | RDATA5 | RDATA6 | RDATA7 | RDATA8 |    |

**Example:**

|        |      |       |          |       |      |       |      |      |  |
|--------|------|-------|----------|-------|------|-------|------|------|--|
| NLRSFD | 100  | 1001  | 1002     | XY    | 1.0  | 2.0   | 0.05 | LONG |  |
|        | 2.1  | 300.0 | 1        | 100.0 | 30.0 | 120.0 | 90.0 | 51   |  |
|        | 0.01 | 0.0   | GRPNAMEx |       |      |       |      |      |  |
|        | 0.0  |       |          |       | 0.0  | 0.    | 0.0  | 0.0  |  |

**Alternate Example with UDS:**

|        |        |      |       |     |     |     |     |   |  |
|--------|--------|------|-------|-----|-----|-----|-----|---|--|
| NLRSFD | 100    | 1001 | 1002  | XY  | 1.0 | 1.0 | 1.0 |   |  |
|        | 1.0    | 1    | 1     | 1.0 | 1.0 | 1.0 | 1.0 | 1 |  |
|        | 1.0    | 1.0  | MYSUB | EXT |     |     |     |   |  |
|        | 1.0E-4 |      |       |     |     |     |     |   |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Nonlinear load set identification number. (Integer > 0; Required)                    |
| GA         | Inner (e.g., damper journal) grid for squeeze film damper. (Integer > 0; Required)   |
| GB         | Outer (e.g., housing) grid for squeeze film damper. (Integer > 0; Required)          |
| PLANE      | Radial gap orientation plane: XY, XZ, or ZX. See Remark 1. (Character; Default = XY) |
| BDIA       | Inner journal diameter. (Real > 0.0; Required)                                       |
| BLEN       | Damper length. (Real > 0.0; Required)  |
| BCLR       | Damper radial clearance. (Real > 0.0; Required)                                      |
| SOLN       | Solution option: LONG or SHORT bearing. (Character; Default = LONG)                  |
| VISCO      | Lubricant dynamic viscosity. (Real > 0.0; Required)                                  |
| PVAPCO     | Lubricant vapor pressure. (Real; Required)   |
| NPORT      | Number of lubrication ports: 1 or 2 (Integer; no Default)                            |
| PRES1      | Boundary pressure for port 1. (Real $\geq$ 0.0; Required if NPORT = 1 or 2)          |

| Descriptor | Meaning  |
|------------|--|
| THETA1     | Angular position for port 1. ( $0.0 \leq \text{Real} > 360.0$ ; Required if NPURT = 1 or 2). See Remark 2.   |
| PRES2      | Boundary pressure for port 2. ( $\text{Real} \geq 0.0$ ; Required if NPURT = 2).   |
| THETA2     | Angular position for port 2. See Remark 2. ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPURT = 2)   |
| NPNT       | Number of finite difference points for damper arc. (Odd Integer $\leq 201$ ; Default = 101)  |
| OFFSET1    | Offset in the SFD direction 1. (Real; Default = 0.0)   |
| OFFSET2    | Offset in the SFD direction 2. (Real; Default = 0.0)   |
| GRPNAM     | The GRPNAM is the name used for the group on the FMS CONNECT SERVICE statement; this is used with an external User Defined Service (UDS). See Remark 6. (Character or Blank) |
| EVALNAME   | Value passed to the external User Defined Service as argument "evalname". (Character or Blank)   |
| RDATAi     | Parameters passed to the external User Defined Service as arguments "parmi". (Real; Default = 0.0). See Remark 7.  |

#### Remarks:

1. The XY, YZ, and ZX planes are relative to the displacement coordinates of GA and GB. The plane coordinates correspond to the NLRSFD directions 1 and 2.

In order to ensure correct results, points GA and GB must satisfy the following three conditions:

- a. They must both be grid points
- b. They must be coincident
- c. They must have parallel displacement coordinate systems

The program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message.

2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the ZX plane.
3. OFFSET1 = Damper housing ID center offset displacement relative to OD center in the horizontal direction. Entered as a positive value for horizontally to the left (negative x-direction) displacement (inches).
4. OFFSET2 = Damper housing ID center offset displacement relative to OD center in the vertical direction. Entered as a positive value for downward (negative y-direction) displacement (inches). Positive entry typically used for -1 g compensation.
5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

6. If GRPNAME is blank, the NLRSFD entry used the values defined in the fields prior to GRPNAME. If GRPNAME contains a valid group (as defined on the FMS CONNECT SERVICE entry), the values in fields BDIA, BLEN, BCLR, SOLN, VISCO, PVAPCO, NPORT, PRES1, THETA1, PRES2, THETA2, NPNT, OFFSET1 and OFFSET2 are ignored and the NLRSFD will obtain its characteristics from an external source via the CONNECT SERVICE architecture.
7. Parameters may be changed by the external service; the changed values will be stored and returned on a subsequent call.
8. The squeeze films damper defined via the CONNECT SERVICE architecture in NLRSFD does not support rotors defined using ROTORSE.

**NLSTEP**

Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101.

Specifies the convergence criteria, step size control and numerical procedure for time/load stepping in SOL 400. For multi-physics, it controls both structural and thermal analysis. Defines analysis preference and control parameters for contact analysis in SOL 101.

There are three groups of data that can be entered through this option:

1. General data which defined parameters that may be used for a variety of simulations. This data is provided by the GENERAL keyword.
2. Selecting the type of procedure used to control the time/load stepping procedure. These procedures are activated by the keywords: LCNT, FIXED, ADAPT, or ARCLN (arc length or continuation method). Only one of the keywords may be chosen in a loadcase.
3. Data associated with the physics type that are activated by the keywords: MECH, HEAT, COUP, and RCHEAT. One can enter as many as necessary.

The NLSTEP is selected by the Case Control Command NLSTEP=ID.

**Format: (For SOL 400)**

| 1         | 2        | 3        | 4       | 5       | 6        | 7       | 8      | 9 | 10 |
|-----------|----------|----------|---------|---------|----------|---------|--------|---|----|
| NLSTEP    | ID       | TOTTIME  | CTRLDEF |         |          |         |        |   |    |
| "GENERAL" | MAXITER  | MINITER  | MAXBIS  | CREEP   |          |         |        |   |    |
| "FIXED"   | NINC     | NO       |         |         |          |         |        |   |    |
| "ADAPT"   | DTINITF  | DTMINF   | DTMAXF  | NDESIR  | SFACT    | INTOUT  | NSMAX  |   |    |
|           | IDAMP    | DAMP     | CRITTID | IPHYS   | LIMTAR   | RSMALL  | RBIG   |   |    |
|           | ADJUST   | MSTEP    | RB      | UTOL    |          |         |        |   |    |
| "ARCLN"   | TYPE     | DTINITFA | MINALR  | MAXALR  |          | NDESIRA | NSMAXA |   |    |
| "HEAT"    | CONVH    | EPSUH    | EPSPH   | EPSWH   | KMETHODH | KSTEPH  |        |   |    |
|           | MAXQNH   | MAXLSH   | LSTOLH  |         |          |         |        |   |    |
| "MECH"    | CONV     | EPSU     | EPSP    | EPSW    | KMETHOD  | KSTEP   | MRCOVN |   |    |
|           | MAXQN    | MAXLS    | LSTOL   | FSTRESS |          |         |        |   |    |
| "COUP"    | HGENPLAS | HGENFRIC |         |         |          |         |        |   |    |
| "RCHEAT"  | SOLVER   | DRLXCA   | ARLXCA  | BALENG  | DAMPC    | GRVCON  | CSGFAC |   |    |
|           | NRLOOP   | OUTINV   | DTIME1  |         |          |         |        |   |    |

Format: (For SOL 101)

| 1      | 2      | 3        | 4        | 5     | 6     | 7     | 8       | 9      | 10 |
|--------|--------|----------|----------|-------|-------|-------|---------|--------|----|
| NLSTEP | ID     |          | CTRLDEF  |       |       |       |         |        |    |
|        | "LCNT" | NINCC    | CONVC    | EPSUC | EPSPC | EPSWC | MAXDIVC | MAXBIS |    |
|        |        | MAXITERC | MINITERC |       |       |       |         |        |    |

Example: (Fixed stepping, 30 increments, total time 4.3, max 5 bisections)

|        |         |     |  |      |  |  |  |  |  |
|--------|---------|-----|--|------|--|--|--|--|--|
| NLSTEP | 10      | 4.3 |  |      |  |  |  |  |  |
|        | GENERAL |     |  | 5    |  |  |  |  |  |
|        | FIXED   | 30  |  |      |  |  |  |  |  |
|        | MECH    | PV  |  | 0.01 |  |  |  |  |  |

Example: (Automatic stepping, total time 4.3. Start out with 2%)

|        |         |      |      |   |   |  |    |  |  |
|--------|---------|------|------|---|---|--|----|--|--|
| NLSTEP | 20      | 4.3  |      |   |   |  |    |  |  |
|        | GENERAL |      |      | 5 |   |  |    |  |  |
|        | ADAPT   | 0.02 | 1.-5 |   | 5 |  | 20 |  |  |

Example: (Using SEVERELY default setting with automatic time stepping)

|        |       |     |          |  |  |  |  |  |  |
|--------|-------|-----|----------|--|--|--|--|--|--|
| NLSTEP | 10    | 4.3 | SEVERELY |  |  |  |  |  |  |
|        | ADAPT |     |          |  |  |  |  |  |  |

Example: (Select the default control parameters for accuracy preference in SOL 101)

|        |    |  |        |  |  |  |  |  |  |
|--------|----|--|--------|--|--|--|--|--|--|
| NLSTEP | 10 |  | LCACCU |  |  |  |  |  |  |
|--------|----|--|--------|--|--|--|--|--|--|

Example: (Four increments, P convergence criterion with error tolerance 1.e-4. The other parameters are defaults for performance preference in SOL 101)

|        |      |   |        |  |       |  |  |  |  |
|--------|------|---|--------|--|-------|--|--|--|--|
| NLSTEP | 10   |   | LCPERF |  |       |  |  |  |  |
|        | LCNT | 4 | P      |  | 1.e-4 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number. (Integer > 0)                |
| TOTTIM     | Total time for the load case. (Real; Default = 1.0) |

| Descriptor | Meaning  |
|------------|--|
| CTRLDEF    | <p>This Keyword is applicable in SOL101 linear static contact analysis or in SOL400 nonlinear static and transient analysis.</p> <p>For SOL400, the values "QLINEAR", "MILDLY", and "SEVERELY" are available along with the option of leaving blank. See Remark 14. for detail description of these three keyword options and the blank option.</p> <p>For SOL101, the values "LCPERF" or "LCACCU" are available along with the option of leaving blank. See Remark 15. for detail description of these two keyword options. If left blank the values as described under the "LCNT" keyword entry will be used.</p> <p>If SOL400 keyword values are present in a SOL101 linear static contact analysis run or SOL101 keyword values are present in a SOL400 nonlinear static run, the keywords are ignored.</p> <p>In SOL400 nonlinear transient analysis, the use of keywords "QLINEAR", "MILDLY", and "SEVERELY" or blank <i>cannot</i> properly set a "FIXED" NINC value or an "ADAPT" DTINITF value so the user <i>still must</i> set whichever one of these keyword entries value is relevant for a transient run.</p> <p>The keywords "QLINEAR", "MILDLY", and "SEVERELY" are ignored for a RC Network heat transfer simulation.</p> |
| "GENERAL"  | Keyword for parameters used for overall analysis.  |
| MAXITER    | Maximum number of iterations allowed for each increment. See Remark 6. (Integer; Default = 10)   |
| MINITER    | Minimum number of iterations needed for each increment. (Integer >0; Default = 1) In contact analysis or CTRLDEF = SEVERELY, Default = 2). When high accuracy is required, it is also recommended to set MINITER = 2.  |
| MAXBIS     | Maximum number of bisections allowed in the current step. (Default = 10). See remark 17. for more information.   |
| CREEP      | Creep will be used in the current load case. If set to 1 creep will be used; if set to 0 creep will not be used (even if creep is defined for a material). (Integer 0 or 1; Default = 0). Adaptive load stepping based upon creep can be flagged through CRITTID below. For enhanced nonlinear elements only.  |
| "FIXED"    | Keyword to indicate that fixed time stepping is to be used. See Remark 1.  |
| NINC       | Number of increments for fixed time stepping. (Integer > 0; Default = 50). The time step of each increment will be TOTTIME/NINC.   |
| NO         | Interval for output. Every NO-th increment will be saved for output. (Integer $\geq 0$ ; Default = 1).   |
| "ADAPT"    | Keyword to indicate that the adaptive load stepping procedure should be used. See Remark 1.  |

| Descriptor | Meaning  |
|------------|--|
| DTINITF    | Initial time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.01). If DTINITF>=DTMAXF then, DTINITF is reset to DTMAXF.<br>If CTRLDEF is set to QLNEAR, the user should set DTINITF equal to TOTTIM.  |
| DTMINF     | Minimum time step defined as fraction of total load step time (TOTTIM). (Real; Default 1e-5).  |
| DTMAXF     | Maximum time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.5). For most nonlinear problems, this should be between 0.05 and 0.2 for dynamic simulations.   |
| NDESIR     | Desired number of iterations per increment. See Remark 2. (Integer; Default = 4)   |
| SFACT      | Factor for increasing time steps due to number of iterations. See Remark 4. (Real; Default = 1.2).   |
| INTOUT     | Output flag. Integer $\geq -1$ . (Default = 0) <ul style="list-style-type: none"> <li>-1 Only the last increment of the step will be output.</li> <li>0 Every computed load increment will be output.</li> <li>&gt; 0 The output will be obtained at INTOUT equally spaced intervals. The time step will be temporarily adjusted if necessary in order to reach these points in time.</li> </ul>                                   |
| NSMAX      | Maximum number of increments in the current load case. (Integer; Default = 99999). The job will stop if this limit is reached.   |
| IDAMP      | Flag for activating artificial damping for static analysis. (Integer, Default = 0). See Remarks 27. <ul style="list-style-type: none"> <li>0 No damping considered</li> <li>4 Artificial damping is always turned on</li> <li>5 Artificial damping is not turned on. But time step is adjusted based on damping energy as 4</li> <li>6 When the time step reaches the minimum value and artificial damping is turned on</li> </ul> |
| DAMP       | Damping ratio. (Real; Default = 2.e-4).  |
| CRITTID    | ID of Bulk Data TABSCTL entry which defines the user criteria to use. See Remark 5. (Integer; Default 0)   |
| IPHYS      | Flag to determine if automatic physical criteria should be added and how analysis should proceed if a user criterion is not satisfied. (Integer; Default = 2) It is recommended to use 1. <ul style="list-style-type: none"> <li>2 Do not add automatic physical criteria; stop when any user criterion is not satisfied</li> </ul>  |

| Descriptor | Meaning  |
|------------|--|
| -2         | Do not add automatic physical criteria; continue when user criteria are not satisfied  |
| 1          | Add automatic physical criteria; stop when any user criterion is not satisfied   |
| -1         | Add automatic physical criteria; continue when any user criterion is not satisfied   |
| LIMTAR     | Enter 0 to treat user criteria as limits, 1 to treat user criteria as targets. (Integer; Default = 0). Only used if a user criterion is given through CRITTID. See Remark 5.                       |
| RSMALL     | Smallest ratio between time step changes due to user criteria. (Real; Default = 0.1)   |
| RBIG       | Largest ratio between time step changes due to user criteria. (Real; Default = 10.0)   |
| ADJUST     | Time step skip factor for automatic time step adjustment. Only for dynamics. See Remark 16. (Integer; Default = 0).  |
| MSTEP      | Number of steps to obtain the dominant period response. See Remark 16. ( $10 \leq \text{Integer} \leq 200$ or = -1; Default = 10).   |
| RB         | Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 16. ( $0.1 \leq \text{Real} \leq 1.0$ ; Default = 0.6).                         |
| UTOL       | Defines tolerance on displacement. ( $.0001 \leq \text{Real} \leq 1.0$ ; Default = 1.0)  |
| "LCNT"     | Keyword to indicate that the contact analysis in SOL101 should be used. See Remark 15.   |
| NINCC      | Number of increments. (Integer > 0; Default = 10 for CTRLDEF= ""; Default=1 for CTRLDEF="LCPERF" or "LCACCU")  |
| CONVC      | Flags to select convergence criteria. (Character="U", "P", "W", "V", or any combination; Default = "PV" for CTRLDEF = "" or "LCPERF"; Default = "UPV" for CTRLDEF= "LCACCU"). See Remark 19.       |
| EPSUC      | Error tolerance for displacement (U) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")  |
| EPSPC      | Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")  |
| EPSWC      | Error tolerance for work (W) criterion. (Real > 0.0; Default = 1.0E-7 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF= "" or "LCACCU")  |
| MAXDIVC    | Limit on probable divergence conditions per iteration before the solution is assumed to diverge. (Integer $\neq 0$ ; Default = 3 for CTRLDEF = "" or "LCPERF"; Default = 5 for CTRLDEF= "LCACCU"). |
| MAXBISC    | Maximum number of bisections allowed for each load increment. (-10 $\leq$ Integer $\leq$ 10; Default = 5)  |
| MAXITERC   | Limit on number of iterations for each load increment. (Integer $\geq 0$ , Default = 25)   |
| MINITERC   | Minimum number of iterations of a load increment. (Integer > 0; Default = 1. In contact analysis, Default = 2).  |

| Descriptor | Meaning   |
|------------|---|
| "ARCLN"    | Keyword to indicate that an arc length load stepping procedure is to be used. It does not support general contact in SOL 400. See Remark 1.                   |
| TYPE       | Constraint type. (Character: "CRIS", "RIKS", or "MRIKS"; Default = "CRIS") See Remark 7.  |
| DTINITFA   | Initial time step defined as a fraction of the load step time (TOTTIME) for the arc-length procedure. (Real; Default .01)                                     |
| MINALR     | Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (0.0 < Real < 1.0; Default = 0.25) |
| MAXALR     | Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (Real > 1.0; Default = 4.0)        |
| NDESIRA    | Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. (Integer > 0; Default = 4) See Remarks 8. and 9.              |
| NSMAXA     | Maximum number of increments in the current load case. (Integer; Default = 1000). The job will stop if this limit is reached                                  |
| "HEAT"     | Keyword for parameters for heat transfer analysis.  |
| CONVH      | Flags to select convergence criteria. (Character = "U", "P", "W", "V", "N", "A" or any combination. (Default UPW)   |
| EPSUH      | Error tolerance for temperature (U) criterion. (Real; Default =1.0E-2)  |
| EPSPH      | Error tolerance for heat flux (P) criterion. (Real; Default =1.0E-2)  |
| EPSWH      | Error tolerance for work (W) criterion. (Real; Default =1.0E-2)   |
| KMETHODH   | Method for controlling stiffness updates. (Character = "PFNT", "AUTO" or "ITER"). Default = "AUTO". See Remark 22.  |
| KSTEPH     | Number of iterations before the stiffness update for the ITER method. (Integer; Default = 1).   |
| MAXQNH     | Maximum number of quasi-Newton correction vectors to be saved on database. (Integer; Default = MAXITER). Not used for PFNT.                                   |
| MAXLSH     | Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.  |
| LSTOLH     | Line Search tolerance. (0.01 < Real < 0.9; Default = 0.5)   |
| "MECH"     | Keyword for parameters for mechanical analysis  |
| CONV       | Flags to select convergence criteria. (Character = "U", "P", "W", "V", "N", "A" or any combination; Default = PV). See Remark 19.                             |
| EPSU       | Error tolerance for displacement (U) criterion. (Real; Default = -0.1)  |
| EPSP       | Error tolerance for residual load (P) criterion. (Real; Default = 0.1)  |
| EPSW       | Error tolerance for work (W) criterion. (Real; Default = 0.1)   |

| Descriptor | Meaning  |
|------------|--|
| KMETHOD    | Method for controlling stiffness updates. (Character = "PFNT" or "ITER"; Default = "PFNT"). See Remark <a href="#">22</a> .  |
| KSTEP      | Number of iterations before the stiffness update for the ITER method. (Integer; Default = 10).   |
| MRCOVN     | Flag to specify if rotations and moments should be included in the convergence testing when CONV is set to UV, UN, PV, PN, UPV or UPN: (Integer; Default = 3) <ul style="list-style-type: none"> <li>0 check on forces, moments, displacements and rotations</li> <li>1 check on forces, moments and displacements</li> <li>2 check on forces, displacements and rotations</li> <li>3 check on forces and displacements</li> </ul> If there is no V and N in the CONV, MRCOVN is ignored and all translational and rotational quantities are used to compute the convergence criteria. |
| MAXQN      | Maximum number of quasi-Newton correction vectors to be saved on the database. (Integer; Default = MAXITER). Not used for PFNT.  |
| MAXLS      | Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.   |
| LSTOL      | Line Search tolerance. ( $0.01 < \text{Real} < 0.9$ ; Default = 0.5). Not used for PFNT.   |
| FSTRESS    | Fraction of effective stress used to limit the sub increment size in material routines ( $0.0 < \text{Real} < 1.0$ ; Default = 0.2) For non material enhanced elements only.   |
| "COUP"     | Keyword for parameters for coupled analysis.   |
| HGENPLAS   | Conversion factor for heat generated due to plasticity. (Real; Default = 0.0)  |
| HGENFRIC   | Conversion factor for heat generated due to friction. (Real; Default = 0.0)  |
| "RCHEAT"   | Keyword to indicate that RC Heat Transfer Analysis is to be performed. See Remark <a href="#">10</a> .   |
| SOLVER     | The Relaxation scheme to be used. See Remark <a href="#">12</a> . (Character; Default = "SNSOR")   |
| DRLXCA     | Diffusion node convergence criterion. See Remark <a href="#">11</a> . ( $\text{Real} \geq 0$ ; Default = 1.e-3)  |
| ARLXCA     | Arithmetic node convergence criterion. See Remark <a href="#">11</a> . ( $\text{Real} \geq 0.0$ ; Default = 1.0e-3 degrees)  |
| BALENG     | Allowable system energy imbalance. See Remark <a href="#">11</a> . ( $\text{Real} \geq 0.0$ ; Default 0.0 energy/time)   |
| DAMPC      | Damping constant. ( $\text{Real} \geq 0.0$ ; Default 0.0 non dimensional)  |
| GRVCON     | Gravitation constant. ( $\text{Real} \geq 0.0$ ; Default $9.81 \text{ length/time}^2$ )  |
| CSGFAC     | Time step control factor. See Remark <a href="#">13</a> . ( $\text{Real} \geq 0.0$ ; Default 1.0 nondimensional)   |
| NRLOOP     | Number of relaxation loops allowed. (Integer $\geq 0$ ; Default 5 loop)  |
| OUTINV     | Output interval. See Remark <a href="#">13</a> . ( $\text{Real} \geq 0.0$ ; Default 60.0 time)   |
| DTIME1     | Time step. See Remark <a href="#">13</a> . ( $\text{Real} \geq 0.0$ ; Default 0.0 time)  |

**Remarks:**

1. Note that the following applies:
  - a. Only one of LCNT, FIXED, ADAPT, ARCLN, or RCHEAT time/load stepping scheme can be used on a specific NLSTEP entry.  
Note that only LCNT may be used in SOL 101. If other selection of load stepping scheme is used, it will be ignored, and the default value of LCNT will be applied
  - b. FIXED or ADAPT may be used for a single physics STEP or for a coupled physics STEP/SUBSTEP.
  - c. If no LCNT, FIXED, ADAPT, or ARCLN appear on a NLSTEP entry, then the default is FIXED with 50 increments.
  - d. The ARCLN method is not supported in contact analysis.
  - e. The ARCLN method should not be used in transient dynamic analysis.
  - f. The ARCLN method should not be used in conjunction with HEAT or COUP.
  - g. The ARCLN method cannot be used with the Intel MKL PARDISO solver.
  - h. In one simulation, one cannot use both HEAT and RCHEAT.
  - i. One cannot use both COUP and RCHEAT.
  - j. The LCNT keyword can only be used in a SOL 101 analysis.
2. The desired number of recycles (NDESIR) can be used in static mechanical, dynamic mechanical (see Remark 16) and heat transfer.
3. In coupled analysis, the time step change is calculated separately for heat and mechanical and the smallest of the two is used.
4. When the time step is increased due to desired number of recycles, the previous time step is multiplied with SFACT. When the time step is decreased the factor is calculated internally based upon the minimum time step.
5. User criteria can be given in the TABSCTL entry via CRITTID. These criteria include rotation, displacements, stresses, strains, creep strains. The time step is decreased if the current value of the value is larger than the user specified limit. If LIMITAR is equal to 1 ("target") it also increases the time step for the next increment if the current value is smaller than the target value given.
6. If MAXITER is given a negative value and the MAXITER number of iterations are obtained, convergence is assumed and the analysis will continue with the next increment.
7. The "ARCLN" entry is applicable to "MECH" analysis only and is ignored for creep analysis. The available constraint types are as follows.

TYPE = "CRIS":

$$\{U_n^i - U_n^O\}^T \{U_n^i - U_n^O\} + w^2 (\mu^i - \mu^O)^2 = \Delta l_n^2$$

TYPE = "RIKS":

$$\{U_n^i - U_n^{i-1}\}^T \{U_n^i - U_n^O\} + w^2 \Delta \mu^i = 0$$

TYPE = "MRIKS":

$$\{U_n^i - U_n^{i-1}\}^T \{U_n^{i-1} - U_n^O\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^O) = 0$$

where:

w = user specified scaling factor (SCALEA)

$\mu$  = load factor

l = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor ( $\mu$ ). As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite, the arc-length method is degenerated to the conventional Newton's method

8. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one increment to the next by:

$$\text{MINALR} \leq \Delta l_{\text{new}} / \Delta l_{\text{old}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use:

$$\text{MINALR} = \text{MAXALR} = 1$$

9. The arc-length l for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous increment ( $I_{\max}$ ) and the number of iterations desired for convergence in the current increment (NDESIRA) as follows:

$$\Delta l_{\text{new}} = \Delta l_{\text{old}} (NDESIRA / I_{\max})^{1/2}$$

10. This entry is required for a non finite element, Resistance-Capacitor network method of analysis for heat transfer.
11. Convergence is determined by the combination of DRLXCA, ARLXCA, and BALENG. DRLXCA and ARLXCA determine if relaxation is met on a node by node basis, rather than a residual vector length.
12. If in Case Control the ANALYSIS=RCNS, then valid values of SOLVER are:

|                 |   |
|-----------------|---|
| SNSOR (Default) | Successive over-relaxation method   |
| STDSTL          | An iterative solver aimed at the fourth root of a quartic for the network equations (good for strong radiation dependence). |

If in Case Control the ANALYSIS=RCNT, then valid values are

|                         |  |
|-------------------------|--|
| SNDUFR<br>(Recommended) | An unconditionally stable, explicit method based on a modified Dufort-Frankel scheme |
| SNFRDL                  | Fast, accurate explicit forward differencing transient method                        |
| FWDBKL                  | Implicit forward/backward differencing Crank Nicolson method                         |
| SNADE                   | Alternating direction explicit method  |
| ATSDUF                  | SNFUFR with automatic time step based on ERRMIN/ERRMAX                               |
| ATSFBK                  | FWDBKL with automatic time step based on ERRMIN/ERRMAX                               |
| SNTSM                   | Weighted implicit forward/backward differencing method                               |
| SNTSM3                  | Weighted implicit forward/backward differencing method                               |
| SNTSM1                  | Weighted implicit forward/backward differencing method                               |
| SNTSM4                  | Weighted implicit forward/backward differencing method                               |

If SOLVER is left blank or set to SNSOR and ANALYSIS=RCNT then internally the RC code will select SNDUFR.

13. About the time step:

- a. The default computed time step ( $DTIMEU$ ) =  $CSGMIN * CSGFAC$ .  $CSGMIN$  is based on the conductance in the model and can be checked in the .sot file. If  $CSGFAC$  is not specified, it is internally set to 1.0.
- b. In a normal sized model,  $CSGMIN$  is usually small enough for the time step which will assure a convergent transient run.
- c.  $CSGFAC$  is used to adjust the time step. It is recommended to determine the best  $CSGFAC$  to the model while maintaining acceptable temperature errors.
- d. If  $OUTPUT < CSGFAC * CSGMIN$  or  $OUTPUT < DTIMEI$ , then  $OUTPUT$  becomes the time step. All the  $OUTPUT$  points are automatically required to be calculated.
- e.  $DTIMEI$  is the forced time step which will ignore any other factors. Sometimes it may lead to inaccurate answers if it is too large.  $DTIMEI$  does not affect the automatic time step solvers.
- f. If the model size is very small,  $CSGMIN$  may be too big for the time step. A small  $CSGFAC$  or  $DTIMEI$  should be used to adjust the time step.
- g.  $CSGFAC * CSGMIN$  or  $DTIMEI$  should be small enough to "catch" any details in time fields, temperature fields or flux arrays.

14. In SOL 400, CTRLDEF entry is only valid for "MECH", "HEAT" and "COUP". The description and default of each option is listed below.

| CTRLDEF  | <p>Keyword to select the default setting of the control parameters. (Character = "QLINEAR", "MILDLY", "SEVERELY". No default.)</p> <p>Stress analysis (NLSTAT, NLTRAN):</p> <pre> QLINEAR : FIXED = 1 inc, ADAPT : initial time ratio = 1.0 (maximum time ratio = 1.0) MILDLY   : FIXED = 10 inc, ADAPT : initial time ratio = 0.1 SEVERELY : FIXED = 50 inc, ADAPT : initial time ratio = 0.01 Blank     : FIXED = 50 inc, ADAPT : initial time ratio = 0.01 </pre> <table border="0"> <thead> <tr> <th></th><th>CONV</th><th>EPSU*</th><th>EPSP</th><th>EPSW</th><th>KMETHOD</th></tr> </thead> <tbody> <tr> <td>QLINEAR</td><td>PV</td><td>0.001</td><td>0.001</td><td>---</td><td>PFNT</td></tr> <tr> <td>MILDLY</td><td>PV</td><td>0.01</td><td>0.01</td><td>---</td><td>PFNT</td></tr> <tr> <td>SEVERELY</td><td>PV</td><td>0.1</td><td>0.01</td><td>---</td><td>PFNT</td></tr> <tr> <td>Blank</td><td>PV</td><td>0.1</td><td>0.1</td><td>---</td><td>PFNT</td></tr> </tbody> </table> <p>[Note] EPSU* is a suggestion when user wants to set CONV=UPV</p> <p>Thermal analysis (HSTAT, HTRAN) :</p> <pre> OLINEAR    : FIXED = 1 MILDLY     : FIXED = 10 SEVERELY   : FIXED = 50 Blank:           : FIXED = 50 </pre> <table border="0"> <thead> <tr> <th></th><th>CONVH</th><th>EPSUH</th><th>EPSPH</th><th>EPSWH</th><th>KMETHOD</th></tr> </thead> <tbody> <tr> <td>OLINEAR</td><td>UPW</td><td>0.001</td><td>0.001</td><td>0.001</td><td>AUTO</td></tr> <tr> <td>MILDLY</td><td>UPW</td><td>0.01</td><td>0.01</td><td>0.01</td><td>AUTO</td></tr> <tr> <td>SEVERELY</td><td>UP</td><td>0.01</td><td>0.01</td><td>---</td><td>AUTO</td></tr> <tr> <td>Blank</td><td>UPW</td><td>0.01</td><td>0.01</td><td>0.01</td><td>AUTO</td></tr> </tbody> </table> <p>For stress analysis, both (1) CTRLDEF and (2) "FIXED" or "ADAPT" have to be set. For heat analysis, both CTRLDEF and "FIXED" have to be set. Otherwise the standard default setting for "FIXED" or "ADAPT" will be set.</p> |       | CONV  | EPSU* | EPSP    | EPSW | KMETHOD | QLINEAR | PV | 0.001 | 0.001 | --- | PFNT | MILDLY | PV | 0.01 | 0.01 | --- | PFNT | SEVERELY | PV | 0.1 | 0.01 | --- | PFNT | Blank | PV | 0.1 | 0.1 | --- | PFNT |  | CONVH | EPSUH | EPSPH | EPSWH | KMETHOD | OLINEAR | UPW | 0.001 | 0.001 | 0.001 | AUTO | MILDLY | UPW | 0.01 | 0.01 | 0.01 | AUTO | SEVERELY | UP | 0.01 | 0.01 | --- | AUTO | Blank | UPW | 0.01 | 0.01 | 0.01 | AUTO |
|----------|---|-------|-------|-------|---------|------|---------|---------|----|-------|-------|-----|------|--------|----|------|------|-----|------|----------|----|-----|------|-----|------|-------|----|-----|-----|-----|------|--|-------|-------|-------|-------|---------|---------|-----|-------|-------|-------|------|--------|-----|------|------|------|------|----------|----|------|------|-----|------|-------|-----|------|------|------|------|
|          | CONV  | EPSU* | EPSP  | EPSW  | KMETHOD |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| QLINEAR  | PV  | 0.001 | 0.001 | ---   | PFNT    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| MILDLY   | PV  | 0.01  | 0.01  | ---   | PFNT    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| SEVERELY | PV  | 0.1   | 0.01  | ---   | PFNT    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| Blank    | PV  | 0.1   | 0.1   | ---   | PFNT    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
|          | CONVH   | EPSUH | EPSPH | EPSWH | KMETHOD |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| OLINEAR  | UPW   | 0.001 | 0.001 | 0.001 | AUTO    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| MILDLY   | UPW   | 0.01  | 0.01  | 0.01  | AUTO    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| SEVERELY | UP  | 0.01  | 0.01  | ---   | AUTO    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |
| Blank    | UPW   | 0.01  | 0.01  | 0.01  | AUTO    |      |         |         |    |       |       |     |      |        |    |      |      |     |      |          |    |     |      |     |      |       |    |     |     |     |      |  |       |       |       |       |         |         |     |       |       |       |      |        |     |      |      |      |      |          |    |      |      |     |      |       |     |      |      |      |      |

15. "LCPERF" specifies the performance preference during analysis, while "LCACCU" prefers accuracy for analysis. These keywords must be defined if the smart contact in SOL 101 default is required. Specification of LCNT keyword is optional. Listed below are default control parameters of contact in SOL 101 if LCNT keyword or some of its fields are not defined. If the CTRLDEF field is blank, default control parameters will be same as those of NLPARM Bulk Data entry.

| CTRLDEF | NINCC | CONVC | EPSUC | EPSPC | EPSWC | MAXDIVC | MAXBISC | MAXITERC | MINITERC |
|---------|-------|-------|-------|-------|-------|---------|---------|----------|----------|
| LCPERF  | 1     | PV    | 1.e-3 | 1.e-3 | 1.e-7 | 3       | 5       | 25       | 2        |
| LCACCU  | 1     | UPV   | 1.e-2 | 1.e-2 | 1.e-2 | 5       | 5       | 25       | 2        |
|         | 10    | PV    | 1.e-2 | 1.e-2 | 1.e-2 | 3       | 5       | 25       | 2        |

The default error tolerances EPSPC and EPSWC are relaxed for the accuracy preference LCACCU. These relaxations are imperative to avoid convergence problems with an additional displacement convergence criterion. The default control parameters are selected based on the test results of typical contact in SOL 101 models. Therefore, these empirical data are only served for the purpose of general convenience. The users can customize any control parameters while performing their analysis.

For some contact models, specifying "LCPERF" or "LCACCU" preference may result in poor convergence. In these cases, the user may increase the number of increments (NINCC) to achieve optimal results.

16. If  $10 \leq \text{MSTEP} \leq 200$  : activating frequency based adaptive time step adjustment with bounds (RB) when specified increment (controlled by ADJUST) converges. It cannot be combined with iteration based adaptive time step adjustment, controlled by NDESIR, and NDESIR is always ignored in this case. If MSTEP = -1: activating frequency based adaptive time step adjustment without bounds (RB). It can be combined with iteration based adaptive time step adjustment controlled by NDESIR. If NDESIR=0, time step is only adjusted after the increment converges.
17. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in  $|\text{MAXBIS}|$  bisection, the analysis is terminated. It is recommended to enter 0.
18. SOL400 only support the displacement error computation with respect to the delta displacement of a load increment for EPSU. Although a positive EPSU is assigned, internally it will be changed into a negative value. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed with respect to the delta energy of a load increment.
19. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if CONV = 'blank' the code will use a default of "UPW" if heat analysis and "PV" if a structural stress analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 20. For A, see Remark 21.
20. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

21. For SOL 400, the convergence checking flag "A" is implemented. "A" means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force

is small, i.e., PVA → UVA . The legal combinations for A are PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example, PVA → UVA , PVA → UNA , etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.

22. The stiffness update strategy is selected in the KMETHOD and KMETHODH field for mechanical and thermal analysis, respectively.
  - AUTO may be used for thermal analysis only. PFNT and ITER may be used in both mechanical and thermal analysis.
  - In the thermal analysis, if the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 17. and 23. for diverging solutions. Available for the heat transfer pass only.
  - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if KSTEP < MAXITER. However, if KSTEP > MAXITER, stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and KSTEP = MAXITER.
  - If the PFNT option is selected, the program will use the Pure Full Newton iteration method.
23. The ratio of energy errors before and after the iteration is defined as divergence rate ( $E^i$ ) , i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If  $E^i \geq 1$  or  $E^i < -10^{12}$  , then  $NDIV = NDIV + 2$

If  $-10^{12} < E^i < -1$  , then  $NDIV = NDIV + 1$

The solution is assumed to diverge when  $NDIV \geq |MAXDIVC|$ . If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIVC is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIVC is negative, the analysis is terminated on the second divergence.

24. Please note that in NLTRAN analysis, the output is also influenced by NLPACK. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP,900,0.2
,fixed,2000,20
,mech,u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

For NO of FIXED in NLSTEP, it is similar to INTOUT of ADAPT.

25. If Modules are present then this entry may only be specified in the main Bulk Data section.
26. NLSTEP entry does not work for creep analysis when using the traditional elements. In SOL 400, NLPARM entry must be used to perform creep analysis with the traditional elements model.
27. Artificial damping takes effect in the following way with different options.

IDAMP=4: Artificial damping is always turned on. Both time stepping and added damping are controlled through the step.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.
- The damping factor to be used is computed based on the estimated damping energy and the estimated total energy for the step.

IDAMP=5: Artificial damping is not turned on. Similar algorithm as IDAMP = 4 except that only the time stepping is controlled based on the damping.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.

IDMAP=6: When the time step reaches the minimum value and the analysis is going to stop prematurely, two attempts are made to avoid a premature exit:

- The increment is repeated with a new time step = 10 tmin and quasi-static damping is added in a manner similar to the IDAMP = 4 scheme.
- If this does not work, the increment is repeated with a new time step = 100 tmin and the process is repeated. Note that once damping is turned on to avoid the premature stop, it remains on for the rest of the step.

**NLSTRAT**

## Strategy Parameters for SOL 600 Nonlinear Structural Analysis

Defines strategy parameters for nonlinear structural analysis used in SOL 600 only. For SOL 600 Heat Transfer, see NLHEATC.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9 | 10 |
|---------|--------|--------|--------|--------|--------|--------|--------|---|----|
| NLSTRAT | ID     | Param1 | Value1 | Param2 | Value2 | Param3 | Value3 |   |    |
|         | Param4 | Value4 | Param5 | Value5 | etc    |        |        |   |    |

**Example:**

|         |         |         |   |       |      |       |     |  |  |
|---------|---------|---------|---|-------|------|-------|-----|--|--|
| NLSTRAT | 501     | CONVTYP | 4 | RESPF | .015 | ALPHA | .05 |  |  |
|         | KNONPOS | 1       |   |       |      |       |     |  |  |

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

|    |   |
|----|---|
| ID | Identification number referenced by a Case Control command with regard to time steps or load steps (such as SUBCASE). If ID = 0, the values entered will be used for Marc increment zero. For the first subcase ID = 1; for the second subcase ID = 2, etc. If there are no subcases in the model, enter ID = 1. If NLSTRAT with ID > 0 is entered and if the ISOLVER option is entered, another NLSTRAT entry with ID = 0 and the same ISOLVER option must be entered. If NLSTRAT is used, there must be an NLSTRAT entry for each subcase. Multiple NLSTRAT entries with the same ID are not allowed. (Integer $\geq 0$ ) |
|----|---|

|        |   |
|--------|---|
| PARAMi | Name of the NLSTRAT parameter. Allowable names are given in <a href="#">Table 9-27</a> . (Character). |
|--------|---|

|      |  |
|------|--|
| VALi | Value of the parameter. See <a href="#">Table 9-27</a> . (Real or Integer) |
|------|--|

Table 9-27 Parameters

| Name    | Description, Type and Value<br>Convergence Criteria  |
|---------|--|
| CONVTYP | Convergence Criteria -- (Integer) - If not set, value will be determined by NLPARM or entry - CONTROL(2,4) The possibilities are:<br>0 -- Convergence based on residuals<br>1 -- Convergence based on displacements<br>2 -- Convergence based on energy<br>4 -- Convergence based on residuals or displacements<br>5 -- Convergence based on residuals and displacements |
| IRELABS | Flag for relative or absolute convergence criteria (Integer) CONTROL(2,5)<br>0 -- Testing is done on relative error<br>1 -- Testing is done on absolute value<br>2 -- Testing is done on relative error testing unless reactions or incremental displacements are below minimum value, in which case absolute tolerance testing is used.                                 |
| RCK1    | Used for Relative Checking - Maximum residual force ratio (maximum allowable value of maximum residual force divided by maximum reaction force) or displacement ratio (maximum allowable value of the change in displacement increment divided by displacement increment) depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,1); Default = 0.1)                            |
| RCK2    | Used for Relative Checking - Maximum residual moment ratio or if autoswitch is on, the rotation ratio depending on CONVTYP (Real $\geq 0$ ; CONTROL(3,2); Default = no checking)   |
| RCK3    | Used for Relative Checking - Minimum reaction force ratio or minimum displacement ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,3); no Default; if 0.0, checking is bypassed or absolute testing is performed)  |
| RCK4    | Used for Relative Checking - Minimum moment ratio or rotation ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,4))   |
| ABCK1   | Used for Absolute Checking - Maximum residual force ratio or displacement ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,5); Default = no checking)  |
| ABCK2   | Used for Absolute Checking - Maximum residual moment ratio or rotation ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,6) - Default = no checking)  |
| MAXDI   | Maximum change in displacement increment divided by displacement increment (Real) Default is no checking (real) - CONTROL (3a-1) - Enter only if CONVTYP is 4 or 5.  |
| MAXRI   | Maximum change in rotational increment divided by rotational increment (Real) Default is no checking (Real) - CONTROL (3a-2) -Enter only if CONVTYP is 4 or 5.   |
| MINDI   | Minimum change in displacement increment divided by displacement increment (real) Default is no checking (Real) - CONTROL (3a-3) - Enter only if CONVTYP is 4 or 5.  |
| MINRI   | Minimum change in rotational increment divided by rotational increment (real) Default is no checking (Real) - CONTROL (3a-4) - Enter only if CONVTYP is 4 or 5.  |

Table 9-27 Parameters (continued)

| Name                     | Description, Type and Value<br>Convergence Criteria  |
|--------------------------|--|
| MAXD                     | Maximum value of displacement increment (Real) Default is no checking (real) - CONTROL (3a-5) - Enter only if CONVTYP is 4 or 5.   |
| MAXR                     | Minimum value of rotational increment (real) Default is no checking (Real) - CONTROL (3a-6) - Enter only if CONVTYP is 4 or 5.   |
| IPRCOV                   | Flag controls printing of convergence. CONTROL(2,9) (Integer; Default = 0 = no print)  |
| AUTOSW                   | Flag to turn on or off Marc's Auto Switch (Integer) Controls switching between convergence testing of residuals and displacements when residuals are small - CONTROL (2,11).<br>0 -- Off (Default unless NLAUTO entry is entered)<br>1 -- On (Default only if NLAUTO entry is entered)   |
| <b>Newton Iterations</b> |  |
| MAXSTEP                  | Maximum number of load steps. CONTROL(2,1) (Integer; Default = 9999)   |
| MAXREC                   | Maximum number of recycle steps per load step (Integer; Default = 3) If set to a negative value, if convergence is not obtained after maxrec recycles, a warning is issued and the analysis proceeds to the next step (not recommended) CONTROL(2,2)   |
| MINREC                   | Minimum number of recycle steps per load step. CONTROL(2,3) (Integer; Default = 0)   |
| IKMETH                   | Newton method (Integer; Default = 1)<br>1= Full Newton,<br>2 = Modified Newton,<br>3 = Newton-Raphson with strain correction,<br>8 = Secant stiffness CONTROL(2,6)   |
| IKUPD                    | Reassembly interval of stiffness and mass. AUTO LOAD (2,2) DYNAMIC CHANGE (2,5) (Integer)  |
| IKNONPOS                 | Solve a non positive definite stiffness (1) or not (0). CONTROL(2,7) For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not. (Integer)   |
| IKINIT                   | Initial stiffness control (Integer) CONTROL (2,10)<br>0 - Normal full contribution<br>1 - For Mooney material, reduced contribution of hydrostatic pressure based on initial stress<br>2 - No initial stiffness<br>3 - Use stress at beginning of increment but not in the last iteration<br>4 - Use only positive stresses in initial stress stiffness (faster than option 0 and is always stable for thin shell structures). |
| <b>General Parameter</b> |  |
| STRAINS                  | Scale factor for strain increments. PARAMETERS(2,1) (Real $\geq 0$ ; Default = 1.0)  |
| PENBOUN                  | Penalty value to enforce certain boundary conditions. PARAMETERS(2,2) (Real)   |

Table 9-27 Parameters (continued)

| Name   | Description, Type and Value<br>Convergence Criteria   |
|--|---|
| FSTRESS                                      | Fraction of the hydrostatic pressure subtracted from the stress tensor in initial stress calculation. PARAMETERS(3,5) (Real)  |
| <b>Load Step or Time Step Control</b>        |   |
| MAXTSC                                       | Maximum number of allowable time step cuts. AUTO LOAD (2,3) (Integer $\geq 0$ )<br>0 - No automatic restart from the previously converged step<br>>1 - Maximum number of time step cutbacks allowed.  |
| <b>Transient Analysis Damping Parameters</b> |   |
| BETA   | Beta parameter used by Newmark-beta procedure. PARAMETERS (2,5) (Real; Default = 0.25)  |
| GAMMA  | Gamma parameter used by Newmark-beta procedure. PARAMETERS(2,6) (Real; Default = 0.5)   |
| GAMMA1                                       | Gamma1 parameter used by Single Step Houbolt procedure. PARAMETERS (2,7) (Real; Default = 1.5)  |
| GAMMA2                                       | Gamma parameter used by Single Step Houbolt procedure. PARAMETERS (2,8) (Real; Default = -0.5)  |
| <b>Solver-Related Parameters</b>             |   |
| ISOLVER                                      | Type of solver (Integer $\geq 0$ ) SOLVER(2,1)<br>0 - Profile Direct Solver<br>2 - Sparse Iterative<br>4 - Sparse Direct<br>6 - Hardware provided direct sparse<br>8 - Multifrontal direct sparse (Default)<br>9 - The CASI iterative solver will be used<br>10 - The mixed direct/iterative solver will be used. The other NLSTRAT entries that apply to solvers 2 and 9 may also be used for solver 10 but are not required.<br>11 - Pardiso direct solver<br>12 - MUMPS parallel direct solver<br>(Note: Solvers that do not require other NLSTRAT entries can be specified by PARAM,MARCSOLV. For solvers 11 or 12 please also see parameters <a href="#">MUMPSOLV</a> and <a href="#">MRTHREAD</a> ) |
| ISYMM  | Nonsymmetric solver option (Integer $\geq 0$ ; Default = 0) SOLVER (2,2)<br>0 - Symmetric solver<br>1 - Non symmetric solver  |
| NONPOS                                       | Nonpositive Definite solver option (Integer $\geq 0$ ; Default = 0) SOLVER (2,3)<br>0 - Error if system is nonpositive-definite<br>1 - Solve nonpositive definite systems if possible<br>For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not.  |

Table 9-27 Parameters (continued)

| Name   | Description, Type and Value<br>Convergence Criteria  |
|--|--|
| MBYTE  | Solver type 6 or 8 memory option (Integer $\geq 0$ ) SOLVER (2,8). Enter the number of 4-byte words in millions to be used if solver type 6 (SGI only) or solver type 8 (all other systems) is to be used. For example, if 96MB is needed, enter 96.   |
| MAXITER  | Maximum number of iterations (Iterative solver only). (Integer; Default = 1000) Enter a negative value if program is to continue even though iterations have not fully converged. SOLVER (3,1)   |
| PREVITER   | Enter 1 if the previous solution is to be used as the initial trial value (Iterative solver only) (Integer $> 0$ ; Default = 0) SOLVER (3,2)   |
| PRECOND  | Preconditioner Option (Iterative solver only) (Integer $> 0$ ) SOLVER (3,3)<br>For the CASI solver (solver 9):<br>1- CASI solver with standard preconditioner<br>0- CASI solver with primal preconditioner<br>For the standard iterative solver (solver 2):<br>3 - Use diagonal preconditioner<br>4 - Use scaled-diagonal preconditioner<br>5 - Use incomplete Cholesky preconditioner |
| CJTOL  | Enter Conjugate Gradient Convergence Tolerance (Iterative solver only) (Real; Default = 0.001) SOLVER (4,1)  |
| <p>Note: Arc Length and Other Parameters for Marc's AUTO INCREMENT Option</p> <p>An NLPCI entry is needed in addition to the options below to trigger the AUTO INCREMENT option. It is usually used for post-buckling problems.)</p> |  |
| AITYPE   | Arc Length Method. AUTO INCREMENT (2,8) (Integer $\geq 0$ ; Default = 3)<br>0 standard load control<br>1 - Crisfield quadratic constraint method<br>2 - Riks/Ramm linear constraint method<br>3 - Modified Riks/Ramm (linear constrain method)<br>4 - Crisfield, switch to modified Riks/Ramm if no real root found  |
| AIMAXCUT   | Maximum number or time step cutbacks. AUTO INCREMENT (2,9) (Integer $\geq 0$ ;<br>Default if not entered = 5)<br>0 - No automatic restart (cutbacks) from previous converged step are allowed<br>$>0$ - Maximum number of load step cutbacks<br>Note for shell buckling problems it is best to set this value to 5 or 10.  |
| AIFRACT  | Fraction of total load increment that is applied in the first cycle of the first increment.<br>AUTO INCREMENT (2,1) (Real)<br>Note this value is not set from NLPARM. It needs to be entered if AUTO INCREMENT is to be used for buckling problems.  |
| AIMAXINC   | Maximum number of increments. For most problems, this value should be entered and set to a large value such as 99999. AUTO INCREMENT (2,2) (Integer $> 0$ )  |

Table 9-27 Parameters (continued)

| Name   | Description, Type and Value<br>Convergence Criteria  |
|--|--|
| AINRECYC   | Desired number of recycles per increment used to increase or decrease load steps. AUTO INCREMENT (2,3) (Integer > 0; Default = 3)  |
| AIMAXF   | Maximum fraction of the total load that can be applied in any increment. It is recommended that for most nonlinear problems, this value be 0.1 or smaller. AUTO INCREMENT (2,4) (Real; Default=0.05 if the model does not have contact and 0.01 if the model has contact). Note for shell buckling problems it is best not to set this value (i.e., leave it as default). If AIMAXF is set to a small value, the problem will probably diverge and/or get bad results. |
| AIARCM   | Maximum arc length multiplier (norm of displacement vector to initial arc length). AUTO INCREMENT (2,5) (Real; Default is fraction of load divided by initial fraction of load)  |
| AITOTT   | Total Time period to be covered, used in conjunction with contact analysis. AUTO INCREMENT (2,6) (Real; Default = 1.0)   |
| AIARC0   | Fraction of the initial arc length to define a minimal arc length. AUTO INCREMENT (2,7) (Real; Default = 0.01)   |
| <b>Fully Coupled Heat-Structural Analysis Controls</b> |  |
| TCHANGE  | Maximum nodal temperature change allowed. CONTROL(4,1) (Real; Default = 20.0)  |
| TEVAL  | Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. CONTROL(4,2) (Real; Default = 100.0)   |
| TERROR   | Maximum error in temperature estimates used for property evaluation. CONTROL(4,3) (Real; Default = 0.0; which bypasses the test)   |
| <b>Contact-Related Parameters</b>                      |  |
| ANG2D  | Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 2D contact (Real; Default = 8.625 degrees) PARAMETERS (3,1)   |
| ANG3D  | Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 3D contact (Real; Default = 20.0 degrees) PARAMETERS (3,2)  |
| <b>Other Parameters</b>                                |  |
| DRILLF   | The factor used to calculate the drilling mode for shell elements (types 22, 75, 138, 139, 140) DRILLF=1.0E-6*K6ROT. The default for DRILLF (0.0001) and K6ROT (100.0) produce the same results. If K6ROT is entered, it will be used for DRILLF unless DRILLF is also entered. DRILLF has precedence over K6ROT. (Real; Default = 0.0001) PARAMETERS (3,6)  |
| REZONEF  | Incremental displacement scale factor after a rezoning increment (Real; Default=1.0). Note that a value of 1.0 improves friction convergence but may result in an inside-out element. PARAMETERS (3,7)   |
| UGAS   | Universal Gas Constant (Real; Default = 8.314 J mol <sup>-1</sup> K <sup>-1</sup> ) PARAMETERS (4,1)   |

Table 9-27 Parameters (continued)

| Name   | Description, Type and Value<br>Convergence Criteria   |
|--|---|
| TOFSET                                       | Offset temperature between user units and absolute zero temperature (Real; Default=273.15 which is correct for Centigrade). If temperature units are Kelvin (K) or Rankine (R), enter a negative value and the temperature offset is set to zero<br>PARAMETERS (4,2)  |
| TWEIGHT                                      | Thermal properties evaluation weight (Real; Default = 0.5) PARAMETERS (4,3)   |
| SPFACT                                       | Surface projection factor for single step Houbolt method (Real; Default = 0.0)<br>PARAMETERS (4,4)  |
| STEFAN                                       | Stefan Boltzman Constant (Real; Default = 5.67051E-8 W / m <sup>2</sup> K <sup>4</sup> ) PARAMETERS (4,5)   |
| PLANKS                                       | Planks second constant. (Real; Default = 14387.69 micro MK PARAMETERS (4,6)   |
| CLIGHT                                       | Speed of light in a vacuum. (Real; Default = 2.9979E14 micro M/s PARAMETERS (4,7)   |
| RAPMAX                                       | Maximum change in the incremental displacement in a Newton-Raphson iteration. (Real; Default = 1.0E30) PARAMETERS (4,8)   |
| FISTIF                                       | Initial friction stiffness for model 6 used in first cycle of an increment to define the friction stiffness matrix in cases where a touching node has a zero normal force and the amount of sliding does not exceed the elastic sticking limit. (Real; Default = 0.0 in which case the program calculates it.) PARAMETERS (5,1) |
| SNGMIN                                       | Minimum value that indicates a singularity if a direct solver is used. (Real; Default = 0.0 in which case the value is set internally by the program.) PARAMETERS (5, 2)  |
| RTMAX  | Maximum change in temperature per iteration in radiation simulations. (Real; Default = 10 times the maximum error in temperature estimate or 100.0) PARAMETERS (5,3)  |
| New Items for Version 2005 r3 and Subsequent |   |
| IASMBL                                       | Assembly flag. If set to 1, the stiffness matrix is assembled each iteration. Note that this switches off the modified Newton-Raphson procedure if chosen. (Integer)  |

Table 9-27 Parameters (continued)

| Name   | Description, Type and Value<br>Convergence Criteria  |
|--------|--|
| INNER  | For some material models, such as damage, cracking, and Chaboche, there is an inner iteration loop to insure accuracy. The maximum number of iterations allowed can be set here. (Integer; Default = 50)   |
| RIGLNK | Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)   |
| RLROTT | Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)<br><br><b>Note:</b> If CONVTYP is 4 or 5, the rigid link rotation tolerance entered here circumvents the corresponding value RIGLNK above. For all versions with PARAM,MARCVR less than 12, the rigid link rotation tolerance if left at 0, is reset to 0.001 radians to ensure backward compatibility for RBE2. In this case, the rigid link rotation tolerance should be set to a negative number to by-pass the check. |
| ENRGCH | Maximum allowable value of the change in energy increment. Enter only if CONVTYP=2. (Real; Default = 0.1)  |

**Remarks:**

1. This entry matches Marc's CONTROL, AUTO LOAD, DYNAMIC CHANGE, PARAMETERS, and SOLVER definitions.
2. NLSTRAT is recognized only when Marc is executed from SOL 600.
3. Correlation between NLSTRAT names and Marc CONTROL entry fields

|              |            |            |             |             |
|--------------|------------|------------|-------------|-------------|
| 2-1 MAXSTEP  | 3-1 RCK1   | 4-1 MAXDI  | 5-1 MAXENRG | 6-1 TCHANGE |
| 2-2 MAXREC   | 3-2 RCK2   | 4-2 MAXRI  |             | 6-2 TEVAL   |
| 2-3 MINREC   | 3-3 RCK3   | 4-3 MINDI  |             | 6-3 TERROR  |
| 2-4 CONVTYP  | 3-4 RCK4   | 4-4 MINRI  |             | 6-4 VOLTMAX |
| 2-5 IRELABS  | 3-5 ABCK1  | 4-5 MAXD   |             |             |
| 2-6 IKMETH   | 3-6 ABCK2  | 4-6 MAXR   |             | 7-1 ESRELER |
| 2-7 IKNONPOS | 4-7 RIGLNK | 4-7 RLROTT |             | 7-2 ESABSER |
| 2-8 Not Used |            |            |             |             |
| 2-9 IPRCONVs |            |            | 5-1 ENRGCH  |             |
| 2-10 IKINT   |            |            |             |             |
| 2-11 AUTOSW  |            |            |             |             |

2-12 IASMBL

2-13 INERR

- Items 6-4,7-1,7-2 are not presently available using SOL 600).
- 3-1 to 3-6 is entered only if CONVTYP=0, 4 or 5.
- 4-1 to 4-6 is entered only if CONVTYP=1, 4 or 5.
- 5-1 is entered only if CONVTYP=2.

#### 4. Correlation between NLSTRAT names and Marc PARAMETERS entry fields

|             |             |             |            |
|-------------|-------------|-------------|------------|
| 2-1 STRAINS | 3-1 ANG2D   | 4-1 UGAS    | 5-1 FISTIF |
| 2-2 PENBOUN | 3-2 ANG3D   | 4-2 TOFSET  | 5-2 SNGMIN |
| 2-3 PFPLAS  | 3-3 RATE0   | 4-3 TWEIGHT | 5-3 RTMAX  |
| 2-4 PFFLUID | 3-4 RATEC   | 4-4 SPFACT  |            |
| 2-5 BETA    | 3-5 FSTRESS | 4-5 STEFAN  |            |
| 2-6 GAMMA   | 3-6 DRILLF  | 4-6 PLANKS  |            |
| 2-7 GAMMA1  | 3-7 REZONEF | 4-7 CLIGHT  |            |
| 2-8 GAMMA2  |             | 4-8 RAPMAX  |            |

#### 5. Correlation between NLSTRAT names and Marc SOLVER entry fields

|             |              |           |
|-------------|--------------|-----------|
| 2-1 ISOLVER | 3-1 MAXITER  | 4-1 CJTOL |
| 2-2 ISYMM   | 3-2 PREVITER |           |
| 2-3 NONPOS  | 3-3 PRECOND  |           |
| 2-8 MBYTE   |              |           |

#### 6. Correlation between NLSTRAT names and Marc AUTO INCREMENT entry fields

|              |              |
|--------------|--------------|
| 2-1 AIFRACT  | 2-6 AITOTT   |
| 2-2 AIMAXINC | 2-7 AIARC0   |
| 2-3 AINRECYC | 2-8 AITYPE   |
| 2-4 AIMAXF   | 2-9 AIMAXCUT |
| 2-5 AIARCM   |              |

#### 7. Correlation between NLSTRAT names and Marc AUTO LOAD entry fields

2-2 IKUPD  
2-3 MAXTSC

8. Correlation between NLSTRAT names and Marc DYNAMIC CHANGE entry fields  
2-5 IKUPD
9. The ISOLVER must be the same for all load cases (and Phase 0) or Marc will abort. It is recommended that all other SOLVER items also be the same for the entire run. If ISOLVER is a value other than 8, NLSTART with ID=0 must be entered with the desired ISOLVER type. Multiply NLSTRAT entries with ID=1, 2, etc. may be used to change other values if desired.
10. ISYMM must be the same for all load cases. All discussions from note 10 apply to ISYMM.

**NOLIN1**

## Nonlinear Load as a Tabular Function

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$\text{Function of displacement: } P_i(t) = S \cdot T(u_j(t)) \text{ or } P_i(f) = S \cdot T(u_j(f)) \quad (9-30)$$

$$\text{Function of velocity: } P_i(t) = S \cdot T(\dot{u}_j(t)) \text{ or } P_i(f) = S \cdot T(\dot{u}_j(f)) \quad (9-31)$$

where  $t$  is time,  $f$  is frequency and  $u_j(t/f)$  and  $\dot{u}_j(t/f)$  are the displacement and velocity at point GJ in the direction of CJ.

Format:

| 1      | 2   | 3  | 4  | 5 | 6  | 7  | 8   | 9 | 10 |
|--------|-----|----|----|---|----|----|-----|---|----|
| NOLIN1 | SID | GI | CI | S | GJ | CJ | TID |   |    |

Example:

|        |    |   |   |     |   |    |   |  |  |
|--------|----|---|---|-----|---|----|---|--|--|
| NOLIN1 | 21 | 3 | 4 | 2.1 | 3 | 10 | 6 |  |  |
|--------|----|---|---|-----|---|----|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Nonlinear load set identification number. (Integer > 0)  |
| GI         | Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0) |
| CI         | Component number for GI. ( $0 < \text{Integer} \leq 6$ ; blank or zero if GI is a scalar or extra point.)  |
| S          | Scale factor. (Real)   |
| GJ         | Grid, scalar, or extra point identification number. (Integer > 0)  |
| CJ         | Component number for GJ according to the following table:  |

| Type of Point | Displacement                   | Velocity                         |
|---------------|--------------------------------|----------------------------------|
| Grid          | $1 \leq \text{Integer} \leq 6$ | $11 \leq \text{Integer} \leq 16$ |
| Scalar        | Blank or zero                  | Integer = 10                     |
| Extra         | Blank or zero                  | Integer = 10                     |

TID Identification number of a TABLEDi entry. (Integer > 0)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.
2. Nonlinear loads may not be referenced on DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN1 entries must be members of the solution set. This means the e-set (EPOINT entry) for modal formulation and the d-set for direct formulation.
4. Nonlinear loads as a function of velocity (9-31) are denoted by components ten greater than the actual component number; i.e., a component of 11 is component 1 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
6. RC network solver does not support NOLIN1 for thermal analysis.
7. When enforced motion is used, the NOLIN1 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
8. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NOLIN2**

## Nonlinear Load as the Product of Two Variables

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = S \cdot X_j(t) \cdot X_k(t) \text{ or } P_i(f) = S \cdot X_j(f) \cdot X_k(f)$$

where  $t$  is time,  $f$  is frequency and  $X_j(t/f)$  and  $X_k(t/f)$  can be either displacement or velocity at points GJ and GK in the directions of CJ and CK.

Format:

| 1      | 2   | 3  | 4  | 5 | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|----|---|----|----|----|----|----|
| NOLIN2 | SID | GI | CI | S | GJ | CJ | GK | CK |    |

Example:

|        |    |   |   |     |   |   |   |  |  |
|--------|----|---|---|-----|---|---|---|--|--|
| NOLIN2 | 14 | 2 | 1 | 2.9 | 2 | 1 | 2 |  |  |
|--------|----|---|---|-----|---|---|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Nonlinear load set identification number. (Integer > 0)  |
| GI         | Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)   |
| CI         | Component number for GI. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if GI is a scalar or extra point.) |
| S          | Scale factor. (Real)   |
| GJ, GK     | Grid, scalar, or extra point identification number. (Integer > 0)  |
| CJ, CK     | Component number for GJ, GK according to the following table:  |

| Type of Point | Displacement                   | Velocity                         |
|---------------|--------------------------------|----------------------------------|
| Grid          | $1 \leq \text{Integer} \leq 6$ | $11 \leq \text{Integer} \leq 16$ |
| Scalar        | Blank or zero                  | Integer = 10                     |
| Extra         | Blank or zero                  | Integer = 10                     |

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR=SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN2 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. GI-CI, GJ-CJ, and G-K-CK may be the same point.

5. Nonlinear loads may be a function of displacement ( $X = u$ ) or velocity ( $X = \dot{u}$ ). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_t = \frac{u_t - u_{t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{t-1}$  is the displacement of GJ-CJ or GK-CK for the previous time step.

6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN2 for thermal analysis.
8. When enforced motion is used, the NOLIN2 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NOLIN3**

## Nonlinear Load as a Positive Variable Raised to a Power

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} S \cdot [X_j(t)]^A, X_j(t) > 0 \\ 0, X_j(t) \leq 0 \end{cases} \quad \text{or} \quad P_i(f) = \begin{cases} S \cdot [X_j(f)]^A, X_j(f) > 0 \\ 0, X_j(f) \leq 0 \end{cases}$$

where  $t$  is time,  $f$  is frequency and  $X_j(t/f)$  may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

| 1      | 2   | 3  | 4  | 5 | 6  | 7  | 8 | 9 | 10 |
|--------|-----|----|----|---|----|----|---|---|----|
| NOLIN3 | SID | GI | CI | S | GJ | CJ | A |   |    |

Example:

|        |   |     |  |      |   |    |      |  |  |
|--------|---|-----|--|------|---|----|------|--|--|
| NOLIN3 | 4 | 102 |  | -6.1 | 2 | 15 | -3.5 |  |  |
|--------|---|-----|--|------|---|----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Nonlinear load set identification number. (Integer > 0)  |
| GI         | Grid, scalar, or extra point identification number at which the nonlinear load is to be applied. (Integer > 0) |
| CI         | Component number for GI. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if GI is a scalar or extra point.)   |
| S          | Scale factor. (Real)   |
| GJ         | Grid, scalar, extra point identification number. (Integer > 0)   |
| CJ         | Component number for GJ according to the following table:  |

| Type of Point | Displacement                   | Velocity                         |
|---------------|--------------------------------|----------------------------------|
| Grid          | $1 \leq \text{Integer} \leq 6$ | $11 \leq \text{Integer} \leq 16$ |
| Scalar        | Blank or zero                  | Integer = 10                     |
| Extra         | Blank or zero                  | Integer = 10                     |

A Exponent of the forcing function. (Real)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN3 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ( $X_j = u_j$ ) or velocity ( $X_j = \dot{u}_j$ ). Velocities are denoted by a component number ten greater than the actual component number; e.g., a component of 16 is component 6 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN4 entry for the negative range of  $X_j(t)$ .
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN3 for thermal analysis.
8. When enforced motion is used, the NOLIN3 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NOLIN4**

## Nonlinear Load as a Negative Variable Raised to a Power

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} -S \cdot [-X_j(t)]^A, X_j(t) < 0 \\ 0, X_j(t) \geq 0 \end{cases} \quad \text{or} \quad P_i(f) = \begin{cases} -S \cdot [-X_j(f)]^A, X_j(f) < 0 \\ 0, X_j(f) \geq 0 \end{cases}$$

where  $t$  is time,  $f$  is frequency and  $X_j(t/f)$  may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

| 1      | 2   | 3  | 4  | 5 | 6  | 7  | 8 | 9 | 10 |
|--------|-----|----|----|---|----|----|---|---|----|
| NOLIN4 | SID | GI | CI | S | GJ | CJ | A |   |    |

Example:

|        |   |   |   |     |     |  |      |  |  |
|--------|---|---|---|-----|-----|--|------|--|--|
| NOLIN4 | 2 | 4 | 6 | 2.0 | 101 |  | 16.3 |  |  |
|--------|---|---|---|-----|-----|--|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Nonlinear load set identification number. (Integer > 0)  |
| GI         | Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0) |
| CI         | Component number for GI. (0 < Integer $\leq$ 6; blank or zero if GI is a scalar or extra point.)           |
| S          | Scale factor. (Real)   |
| GJ         | Grid, scalar, or extra point identification number. (Integer > 0)  |
| CJ         | Component number for GJ according to the following table:  |

| Type of Point | Displacement              | Velocity                    |
|---------------|---------------------------|-----------------------------|
| Grid          | $1 \leq$ Integer $\leq 6$ | $11 \leq$ Integer $\leq 16$ |
| Scalar        | Blank or zero             | Integer = 10                |
| Extra         | Blank or zero             | Integer = 10                |

A Exponent of forcing function. (Real)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN4 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ( $X_j = u_j$ ) or velocity ( $X_j = \dot{u}_j$ ). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN3 entry for the positive range of  $X_j(t)$ .
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN4 for thermal analysis.
8. When enforced motion is used, the NOLIN4 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NSM****Non Structural Mass Entry by ID**

Defines a set of non structural mass.

**Format:**

| 1   | 2   | 3     | 4      | 5     | 6  | 7     | 8  | 9     | 10 |
|-----|-----|-------|--------|-------|----|-------|----|-------|----|
| NSM | SID | TYPE  | ID     | VALUE | ID | VALUE | ID | VALUE |    |
|     | ID  | VALUE | -etc.- |       |    |       |    |       |    |

**Example:**

|     |   |        |    |      |  |  |  |  |  |
|-----|---|--------|----|------|--|--|--|--|--|
| NSM | 3 | PSHELL | 15 | .022 |  |  |  |  |  |
|-----|---|--------|----|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of non structural mass set. (Integer > 0)  |
| TYPE       | Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character) |
| ID         | Property or Element ID. (Integer > 0)  |
| VALUE      | NSM value (Real)   |

**Remarks:**

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is  $1000 \cdot ID + i$ , where  $i = 1$  to number of harmonics.
3. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

**NSM1****Alternate Form for NSM Entry**

Defines non structural mass entries by VALUE, ID list.

**Format:**

| 1    | 2   | 3    | 4     | 5    | 6  | 7  | 8  | 9  | 10 |
|------|-----|------|-------|------|----|----|----|----|----|
| NSM1 | SID | TYPE | VALUE | ID   | ID | ID | ID | ID |    |
|      | ID  | ID   | ID    | etc. | -  |    |    |    |    |

**Example:**

|      |   |         |      |      |      |      |      |    |  |
|------|---|---------|------|------|------|------|------|----|--|
| NSM1 | 3 | ELEMENT | .044 | 1240 | 1500 | THRU | 1600 | BY |  |
|      | 2 | 2440    | THRU | 2560 |      |      |      |    |  |

Alternate Form and Example(s), applicable when NASTRAN SYSTEM(444)=0 (IFPSTAR=NO):

(All must be in FIELD 5 and non continuation is allowed)

|      |     |      |       |     |      |    |    |   |  |
|------|-----|------|-------|-----|------|----|----|---|--|
| NSM1 | SID | TYPE | VALUE | ID  | THRU | ID |    |   |  |
| NSM1 | SID | TYPE | VALUE | ALL |      |    |    |   |  |
| NSM1 | SID | TYPE | VALUE | ID  | THRU | ID | BY | N |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of non structural mass set. (Integer > 0)  |
| TYPE       | Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character) |
| VALUE      | NSM value (Real)   |
| ID         | Property or Element ID. (Integer > 0 or “ALL” or “THRU” or “BY” or N (the BY increment))   |

**Remarks:**

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is  $1000 \cdot ID + i$ , where  $i = 1$  to number of harmonics.
3. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:

NSM1,12,PCOMP,0.045,ALL

would for example get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLS would have .045 added to their nonstructural mass.

4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

**NSMADD****Non Structural Mass Set Combination**

Defines non structural mass as the sum of the sets listed.

**Format:**

| 1      | 2   | 3  | 4   | 5    | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|-----|------|----|----|----|----|----|
| NSMADD | SID | S1 | S2  | S3   | S4 | S5 | S6 | S7 |    |
|        | S8  | S9 | S10 | etc. | -  |    |    |    |    |

**Example(s):**

|        |    |    |    |    |    |    |    |    |  |
|--------|----|----|----|----|----|----|----|----|--|
| NSMADD | 3  | 17 | 18 | 19 | 20 | 22 | 23 | 24 |  |
|        | 25 | 26 | 27 | 28 |    |    |    |    |  |
| NSMADD | 3  | 29 | 40 | 50 | 55 |    |    |    |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of non structural mass set. (Integer > 0)  |
| Si         | Identification numbers of non structural mass sets defined via NSM, NSML, NSM1, and NSML1 entries. (Integer > 0; SID ≠ Si) |

**Remarks:**

1. The non structural mass sets must be selected with the Case Control command NSM = SID.
2. No Si may be the identification number of a non structural mass set defined by another NSMADD entry.
3. NSMADD entries take precedence over NSM, NSML, NSM1 or NSML1 entries. If both have the same set ID, only the NSMADD entry will be used.
4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

**NSML****Lumped Non Structural Mass Entry by ID**

Defines a set of lumped non structural mass.

Format:

| 1    | 2   | 3     | 4      | 5     | 6  | 7     | 8  | 9     | 10 |
|------|-----|-------|--------|-------|----|-------|----|-------|----|
| NSML | SID | TYPE  | ID     | VALUE | ID | VALUE | ID | VALUE |    |
|      | ID  | VALUE | -etc.- |       |    |       |    |       |    |

Example:

|      |   |        |    |     |  |  |  |  |  |
|------|---|--------|----|-----|--|--|--|--|--|
| NSML | 3 | PSHELL | 15 | .66 |  |  |  |  |  |
|------|---|--------|----|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of non structural mass set. (Integer > 0)  |
| TYPE       | Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character) |
| ID         | Property or Element ID. (Integer > 0)  |
| VALUE      | A lumped mass value to be distributed. (Real)  |

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is  $NSM = \text{VALUE}/\sum_{\text{elements}} \text{Area}$  and for Line elements the calculation is  $NSM = \text{VALUE}/\sum_{\text{elements}} \text{Length}$ .
3. Non structural mass sets must be selected with Case Control command NSM = SID.
4. This entry is not allowed for the CCONEAX element.
5. This entry will cause an equivalent NSM entry to be generated using the computed value for NSM.
6. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.
7. Undefined property/element IDs are ignored.

**NSML1**

## Alternate Form for NSML Entry

Defines lumped non structural mass entries by VALUE, ID list.

**Format:**

| 1     | 2   | 3    | 4     | 5    | 6  | 7  | 8  | 9  | 10 |
|-------|-----|------|-------|------|----|----|----|----|----|
| NSML1 | SID | TYPE | VALUE | ID   | ID | ID | ID | ID |    |
|       | ID  | ID   | ID    | etc. | -  |    |    |    |    |

**Example:**

|       |   |         |      |      |      |  |  |  |  |
|-------|---|---------|------|------|------|--|--|--|--|
| NSML1 | 3 | ELEMENT | .044 | 1240 | 1500 |  |  |  |  |
|-------|---|---------|------|------|------|--|--|--|--|

**Alternate Form and Example(s):**

|       |      |      |       |    |      |      |    |      |  |
|-------|------|------|-------|----|------|------|----|------|--|
| NSML1 | SID  | TYPE | VALUE | ID | THRU | ID   | ID | THRU |  |
|       | ID   | ID   | THRU  | ID | ID   | THRU | ID | ID   |  |
|       | THRU | ID   | ...   |    |      |      |    |      |  |

|       |      |        |      |       |      |       |  |  |  |
|-------|------|--------|------|-------|------|-------|--|--|--|
| NSML1 | 15   | PSHELL | .067 | 1240  | THRU | 1760  |  |  |  |
|       | 2567 | THRU   | 2568 | 35689 | THRU | 40998 |  |  |  |
|       | 76   | THRU   | 300  |       |      |       |  |  |  |

|       |     |      |       |    |      |     |    |   |  |
|-------|-----|------|-------|----|------|-----|----|---|--|
| NSML1 | SID | TYPE | VALUE | ID | THRU | ID  | BY | N |  |
|       | ID  | THRU | ID    | BY | N    | ... |    |   |  |

|       |       |        |       |       |      |       |      |      |  |
|-------|-------|--------|-------|-------|------|-------|------|------|--|
| NSML1 | 3     | PSHELL | .067  | 1240  | THRU | 1760  | 1763 | 1764 |  |
|       | 2567  | THRU   | 2568  | 35689 | TO   | 40999 | BY   | 2    |  |
|       | 76666 | 76668  | 79834 |       |      |       |      |      |  |

(ALL must be in FIELD 5 and no continuation is allowed)

|       |     |       |       |     |  |  |  |  |  |
|-------|-----|-------|-------|-----|--|--|--|--|--|
| NSML1 | SID | TYPE  | VALUE | ALL |  |  |  |  |  |
| NSML1 | 59  | PTUBE | .0123 | ALL |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of non structural mass set. (Integer > 0)  |
| TYPE       | Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character) |

| Descriptor | Meaning  |
|------------|--|
| VALUE      | A lumped mass value to be distributed (Real)   |
| ID         | Property or Element ID. (Integer > 0 or "ALL" or "THRU" or "TO" or "BY" or N (the BY increment)) |

**Remarks:**

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is  $NSM = VALUE / \sum_{elements} \text{Area}$  and for Line elements the calculation is  $NSM = VALUE / \sum_{elements} \text{Length}$ .
3. For NSML1 entries with multiple "THRU" and "THRU,BY" and "ID" lists or any such combination of entries, the  $NSM = VALUE / \sum_{elements} \text{Area}$  and for Line elements the calculation is  $NSM = VALUE / \sum_{elements} \text{Length}$  is based on the individual parent card plus all continuation entries. If an element appears more than once in these multiple combinations, its area or length will be used multiple times in the sum.
4. Nonstructural mass sets must be selected with Case Control command NSM=SID.
5. This entry is not allowed for the CCONEAX element.
6. PBEAML and PCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:  
`NSML1,12,PCOMP,1.35,ALL`  
would, for example, get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLS would have a mass of 1.35 added to their nonstructural mass.
7. The ELSUM Case Control command will give a summary of both structural and non structural mass by element or property type.
8. With the "THRU" and "THRU", "BY" forms, blanks fields are allowed for readability. Any combination of a list of IDs and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing IDs. That is the list of IDs in a THRU range need not be continuous.
9. Undefined property/element IDs are ignored.

**NTHICK****Nodal Thickness Values - SOL 600**

Defines nodal thickness values for beams, plates and/or shells. This is the Marc's nodal thickness option used in SOL 600 only.

**Format:**

| 1      | 2   | 3   | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|-------|---|---|---|---|---|----|
| NTHICK | ID1 | ID2 | THICK |   |   |   |   |   |    |

**Example:**

|        |     |     |       |  |  |  |  |  |  |
|--------|-----|-----|-------|--|--|--|--|--|--|
| NTHICK | 151 | 180 | 0.255 |  |  |  |  |  |  |
|--------|-----|-----|-------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID1        | First Nodal ID to which the thickness applies. (Integer > 0)                                 |
| ID2        | Last Nodal ID to which the thickness applies. (Integer; Default = EID1)                      |
| THICK      | Thickness for all beam, plate or shell elements connecting the nodes specified. (Real > 0.0) |

**Remarks:**

1. The option allows specification of beam, plate and/or shell thickness on a nodal basis. Thickness values specified on property entries overrides values specified by this entry.
2. For all elements including composite elements, nodal thickness is the total thickness.
3. Discontinuities must be modeled using property entries.

**OMIT****Omitted Degrees-of-Freedom**

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

**Format:**

| 1    | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|------|-----|----|-----|----|-----|----|-----|----|----|
| OMIT | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|      |    |   |    |      |  |  |   |   |  |
|------|----|---|----|------|--|--|---|---|--|
| OMIT | 16 | 2 | 23 | 3516 |  |  | 1 | 4 |  |
|------|----|---|----|------|--|--|---|---|--|

| Descriptor | Meaning   |
|------------|---|
| IDi        | Grid or scalar point identification number. (Integer > 0)   |
| Ci         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.) |

**Remarks:**

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “[Degree-of-Freedom Sets](#)” on page 1557 for a list of these entries.
2. Up to 24 degrees-of-freedom may be specified on a single entry.
3. In many cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.
4. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

**OMIT1****Omitted Degrees-of-Freedom, Alternate Form 1**

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

**Format:**

| 1     | 2  | 3  | 4   | 5      | 6  | 7  | 8  | 9  | 10 |
|-------|----|----|-----|--------|----|----|----|----|----|
| OMIT1 | C  | G1 | G2  | G3     | G4 | G5 | G6 | G7 |    |
|       | G8 | G9 | G10 | -etc.- |    |    |    |    |    |

**Example:**

|       |   |   |   |   |    |   |   |   |  |
|-------|---|---|---|---|----|---|---|---|--|
| OMIT1 | 3 | 2 | 1 | 3 | 10 | 9 | 6 | 5 |  |
|       | 7 | 8 |   |   |    |   |   |   |  |

**Alternate Format and Example:**

|       |   |    |        |     |  |  |  |  |  |
|-------|---|----|--------|-----|--|--|--|--|--|
| OMIT1 | C | G1 | "THRU" | G2  |  |  |  |  |  |
| OMIT1 | 0 | 17 | THRU   | 109 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.) |
| Gi         | Grid or scalar point identification number. (Integer > 0; for "THRU" option, G1 < G2.)  |

**Remarks:**

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See "[Degree-of-Freedom Sets](#)" on page 1557 for a list of these entries.
2. If the alternate format is used, not all points in the range G1 through G2 have to be defined. Undefined points will collectively produce a warning message but will otherwise be ignored.
3. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

**OMITAX**

## Omitted Conical Shell Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

**Format:**

| 1      | 2    | 3    | 4  | 5    | 6    | 7  | 8 | 9 | 10 |
|--------|------|------|----|------|------|----|---|---|----|
| OMITAX | RID1 | HID1 | C1 | RID2 | HID2 | C2 |   |   |    |

**Example:**

|        |   |   |   |   |   |   |  |  |  |
|--------|---|---|---|---|---|---|--|--|--|
| OMITAX | 2 | 6 | 3 | 4 | 7 | 1 |  |  |  |
|--------|---|---|---|---|---|---|--|--|--|

| Descriptor       | Meaning  |
|------------------|--|
| RID <sub>i</sub> | Ring identification number. (Integer > 0)  |
| HID <sub>i</sub> | Harmonic identification number. (Integer $\geq 0$ )  |
| C <sub>i</sub>   | Component number(s). (Any unique combination of the Integers 1 through 6 with no embedded blanks.) |

**Remarks:**

1. OMITAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may be specified on this entry.
3. Degrees-of-freedom appearing on OMITAX entries may not appear on MPCAX, SUPAX, or SPCAX entries.

**OUTPUT**

## Output Control for Adaptive Analysis

Output control for p-adaptive analysis.

Format:

| 1  | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--|-----|---|---|---|---|---|---|---|----|
| OUTPUT   | SID |   |   |   |   |   |   |   |    |
| ELSET=n, cmd1=(option1, option2, etc.), cmd2=(option1, etc.), etc. |     |   |   |   |   |   |   |   |    |
| ELSET=m, -etc.-  |     |   |   |   |   |   |   |   |    |

Example:

|  |     |  |  |  |  |  |  |  |  |
|--|-----|--|--|--|--|--|--|--|--|
| OUTPUT   | 127 |  |  |  |  |  |  |  |  |
| ELSET=12, DISP=PRINT, STRESS=(PRINT,PUNCH), STRAIN=PUNCH |     |  |  |  |  |  |  |  |  |
| ELSET=42, STRESS=PRINT,BY=1                              |     |  |  |  |  |  |  |  |  |

| Descriptor | Meaning  | Type        | Default        |
|------------|--|-------------|----------------|
| SID        | DATAREC ID selected in Case Control.<br>See Remark 1.  | Integer > 0 | Required       |
| ELSET      | ID of SET entry containing sets of elements with results that will be processed. See Remark 1.   | Integer > 0 | 999999         |
| cmdi       | Output commands.   | See below   |                |
| optioni    | Specifies one or more of the following output options. The following options may be specified in any order without regard to field boundaries. | See below   | Required       |
| DISP       | Request for calculating displacements.<br>See Remark 1a.   | Character   | DISP = PRINT   |
| VELO       | Request for calculating velocities. See Remarks 1a., 7., and 8.  | Character   | VELO = NONE    |
| ACCE       | Request for calculating accelerations. See Remarks 1a., 7., and 8.   | Character   | ACCE = NONE    |
| STRESS     | Request for calculating stresses. See Remark 1a.   | Character   | STRESS = PRINT |
| STRAIN     | Request for calculating strains. See Remark 1a.  | Character   | STRAIN = NONE  |
| FORCE      | Request to output forces/length in shell elements or forces in beam elements.  | Character   | FORCE = NONE   |
| ERROR      | Request for error estimate table. See Remark 1a.   | Character   | ERROR = PRINT  |

| Descriptor | Meaning  | Type             | Default      |
|------------|--|------------------|--------------|
| PVAL       | Request for new pval values. See Remark 1a.  | Character        | PVAL = PRINT |
| LAST       | Request to print results of last analysis in an adaptive analysis. See Remark 1b.  | Character        | LAST = YES   |
| BY         | Request to print intermediate results in an adaptive analysis. See Remark 1c.      | Integer $\geq 0$ | BY = 0       |
| FIRST      | Request to print results of first analysis in an adaptive analysis. See Remark 1b. | Character        | FIRST = YES  |

#### Remarks:

1. ELSET = n indicates the start of a new set of commands. Commands appearing after ELSET apply only to elements in SET n.
  - a. For cmdi: DISP, VELO, ACCE, STRESS, STRAIN, FORCE, ERROR, and PVAL the allowable optioni are PRINT, PLOT, PUNCH, REAL, IMAG, PHASE, or NONE. If more than one option is desired, enclose in parentheses; e.g., DISP = (PRINT, PUNCH).
  - b. For cmdi: STRAIN, the allowable optioni are STRCUR, FIBER, PRINT, PLOT, PUNCH, or NONE. If more than one option is desired, enclose in parentheses; e.g., STRAIN = (FIBER, PRINT, PUNCH). The options STRCUR and FIBER are for shell elements, they are ignored for other elements. For STRCUR membrane strain and curvature are output, for FIBER, strains in the fibers Z1 and Z2 are output. Z1 and Z2 are specified on the PSHELL Bulk Data entry. The Default is STRCUR. Either STRCUR or FIBER should be specified, but not both.
  - c. For cmdi: FIRST and LAST, the allowable optioni are YES and NO. For example, "FIRST = YES".
  - d. For cmdi: BY, the allowable optioni is an integer greater than or equal to 0. optioni specifies that cmdi will be processed at every adaptive cycle that is a multiple of optioni.
2. Only the output (displacements, stresses, etc.) requested will be either printed or stored for postprocessing. optioni = PRINT (for print in F06 file), PUNCH (for print in punch file), and PLOT (for calculation to be used by postprocessing but not printing) can be used in any combination. For example, DISP = (PRINT), STRESS = (PRINT,PUNCH) will result in printing of displacement data in the F06 file and printing of the stress data in both the F06 file and the punch file.
3. If an element is specified in more than one ELSET = n, then the union of all commands will be performed on that element.
4. SET = 999999 is a reserved set that includes all elements.
5. A command and its options must be specified entirely on the same entry.
6. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
7. VELO and ACCE output commands are only available for transient and frequency response problems.

8. For modal transient and modal frequency analyses with the default matrix data recovery method, requests of velocity or acceleration output must be accompanied by the displacement request for the same set of elements (ELSET). The complex output formats of displacements, velocities, and accelerations are specified by the REAL, IMAG, or PHASE option of the DISP command.
9. The REAL or IMAG option (the default) is used to request rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
10. The PHASE option is used to request polar format (magnitude and phase) of complex output. Phase output is in degrees.

## OUTRCV

## Output Options for p-elements

Defines options for the output of displacements, stresses, and strains of p-elements.

Format:

| 1      | 2       | 3     | 4 | 5 | 6 | 7       | 8 | 9 | 10 |
|--------|---------|-------|---|---|---|---------|---|---|----|
| OUTRCV | SID     | SETID |   |   |   |         |   |   |    |
|        | OPTION1 |       |   |   |   | OPTION2 |   |   |    |

Examples:

|        |       |     |  |  |  |            |  |  |  |
|--------|-------|-----|--|--|--|------------|--|--|--|
| OUTRCV | 150   | 160 |  |  |  |            |  |  |  |
|        | CID=2 |     |  |  |  | VIEW=3*3*9 |  |  |  |

|        |   |   |  |  |  |  |  |  |  |
|--------|---|---|--|--|--|--|--|--|--|
| OUTRCV | 3 | 5 |  |  |  |  |  |  |  |
|--------|---|---|--|--|--|--|--|--|--|

| Descriptor | Meaning  | Type                            | Default      |
|------------|--|---------------------------------|--------------|
| SID        | Identification number. SID is selected by the OUTRCV Case Control command.   | Integer > 0                     | Required     |
| SETID      | Set identification number of a SET Case Control command that appears after the SETS DEFINITION or OUTPUT(POST) command.  | Integer > 0                     | 999999       |
| OPTIONi    | Specifies one or more of the following options. The following options may be specified in any order without regard to field boundaries.  | See CID and VIEW below          |              |
| CID        | Specifies the output coordinate system for all stresses, strains, and displacements, except displacements at points defined by GRID entries. CID = 0 specifies the basic coordinate system; and CID = id specifies a CORDij entry. See Remark 4. | Integer $\geq 0$                | CID = 0      |
| VIEW       | Specifies the intervals for displacement, stress, and strain $\xi \cdot \eta \cdot \zeta$ is the number of subdivisions in $\xi \cdot \eta \cdot \zeta$ of the element's output recovery parametric system. See Remark 5.                        | Three Integers separated by “*” | VIEW = 3*3*3 |

| Descriptor | Meaning  | Type   | Default    |
|------------|--|--|------------|
| PROJ       | Specifies the orientation of a convective coordinate system for shells. PROJ = X specifies the coordinate axis in the CID system which is projected to define the x-axis of the convective coordinate system (tangent system) for shells and beams. Ignored for solids. A minus sign specifies the reverse direction. See Remarks 5. and 10. for more details.   | Character; F,<br>X, Y, Z, -X, -Y, -Z               | PROJ = X   |
| NORMAL     | Specifies the positive direction of the outward normal for shell elements in the CID coordinate system. For NORMAL=R, the positive direction of the outward normal is the exiting arrow side of a radius vector from the origin of the CID system to the element center. For NORMAL = E, the positive direction of the outward normal is the z-axis of the element coordinate system. A minus sign specifies the reverse direction. See Remark 10. for more details. | Character; R,<br>E, -R, -E, X, Y, Z,<br>-X, -Y, -Z | NORMAL = R |
| THETA      | Angle in degrees which rotates the convective system defined with CID and PROJ. THETA is measured in the tangent plane of the shell from the projected axis (selected in PROJ) to the x-axis of the final output coordinate system. For shell elements only.   | Real   | THETA = 0. |

**Remarks:**

1. OUTRCV is intended for p-elements only and specifies the coordinate system and density used for displacement, stress, strain, and force output. OUTRCV is used only for output and has no effect on the solution.
2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. Sets referenced by SETID are defined on the SET command after the SETS DEFINITION or OUTPUT(POST) command. Any p-element not referenced by the SET = SETID Case Control command will use the defaults listed above for CID and VIEW.
4. If an element is referenced by more than one OUTRCV entry then a warning message will be issued and the last OUTRCV will be applied to the element.

5.  $\xi^*\eta^*j$  represents the  $\xi$ ,  $\eta$  and  $\zeta$  subdivisions in the solid element's output recovery parametric system. Both "\*" delimiters are required.  $\eta$  is ignored for the CPENTA and CTETRA element and  $\xi$  is ignored for the CTETRA, CQUAD4, and CTRIA3 element.
6. The elements referenced by the SET = SETID command are labeled in the stress output as VUHEXA, VUPENTA, VUTETRA, VUQUAD, VUTRIA, and VUBEAM. They may be renamed via the PARAM,VUHEXA; PARAM,VUPENTA; PARAM,VUTETRA; PARAM,VUQUAD; PARAM,VUTRIA; and PARAM,VUBEAM entries.
7. Only one OUTRCV Case Control command is allowed. Multiple OUTRCV Bulk Data entries with the same SID are allowed to specify multiple element sets with different output coordinate systems.
8. The displacement output at locations defined by the GRID Bulk Data entry are determined by the CD value located on the GRID Bulk Data entry.
9. For p-version shell elements, the default output coordinates system is the convective coordinate system tangent to the shell mid surface. The x-axis of the convective system is the projected x-axis of the basic system. For p-version beam elements, the output system is the convective coordinate system tangent to the beam axis, oriented from grid A to grid B, as specified on the CBEAM entry.
10. The PROJ and NORMAL options for shells are described in the following list.

|                |  |
|----------------|--|
| PROJ           | Defines the orientation of the output coordinate system for stresses, strains and forces in shell elements. The reference system for PROJ is the CID coordinate system.  |
| PROJ = F       | Stresses, strains and forces of shells are output in the fixed CID. This option should be used if a postprocessor requires the results in terms of 3D vectors or tensors. For example, stress tensors with 6 components. The option does not produce output in the f06 file. |
| PROJ = X, Y, Z | The x- or y- or z-axis of the CID system is projected on to the shell tangent plane, the projected vector defines the x-axis of the convective coordinate system for output of stresses, strains and forces.   |
| NORMAL         | Specifies the positive normal direction of the output coordinate system for stresses, strains, and forces in shell elements. The reference system for NORMAL is the CID coordinate system.   |
| NORMAL = R     | The positive direction of the normal is the exiting arrow of the position vector from the origin of the CID system to the element center.  |
| NORMAL = E     | The positive direction of the normal is the z-axis of the element coordinate system.   |
| NORMAL = X     | The positive direction of the outward normal is the exiting arrow of the x-axis.   |
| NORMAL = Y,Z   | See above.   |

## Entries P

### PAABSF

Frequency-Dependent Absorbers Element Property

#### Format:

| 1      | 2   | 3      | 4      | 5 | 6 | 7 | 8 | 9    | 10 |
|--------|-----|--------|--------|---|---|---|---|------|----|
| PAABSF | PID | TZREID | TZIMID | S | A | B | K | RHOC |    |

#### Example:

|        |    |    |    |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|
| PAABSF | 44 | 38 | 47 |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number that matches the identification number of the corresponding CAABSF entry. (Integer > 0)   |
| TZREID     | Identification number of a TABLEDi entry that defines the resistance as a function of frequency. The real part of the impedance. See Remark 1. (Integer $\geq 0$ or blank)     |
| TZIMID     | Identification number of a TABLEDi entry that defines the reactance as a function of frequency. The imaginary part of the impedance. See Remark 1. (Integer $\geq 0$ or blank) |
| S          | Impedance scale factor. (Real; Default = 1.0)  |
| A          | Area factor when 1 or 2 grid points are specified on the CAABSF entry. (Real $> 0.0$ ; Default = 1.0)  |
| B          | Equivalent structural damping coefficient. See Remark 1. (Real $\geq 0.0$ ; Default = 0.0)   |
| K          | Equivalent structural stiffness coefficient. See Remark 1. (Real $\geq 0.0$ ; Default = 0.0)   |
| RHOC       | Constant used in data recovery for calculating an absorption coefficient. RHO is the media density, and C is the speed of sound in the media. (Real; Default = 1.0)            |

#### Remarks:

- At least one of the four fields TZREID, TZIMID, B, or K must be specified.
- If only one grid point is specified on the CAABSF entry, then the impedance  $Z(f) = Z_R + iZ_I$  is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area.  
 $Z_R(f) = TZREID(f) + B$  and  $Z_I(f) = TZIMID(f) - K/(2\pi f)$ .
- The resistance represents a damper quantity B. The reactance represents a quantity of the type  $(\omega M - K/\omega)$ . The impedance is defined as  $Z = p/u$  where  $p$  is the pressure and  $u$  is the velocity.
- The impedance scale factor S is used in computing element stiffness and damping terms as:

$$k = \frac{A}{S} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

$$b = \frac{A}{S} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

The value of  $(Z_R^2 + Z_I^2)$  must be greater than machine epsilon--a machine dependent constant in the neighborhood of 1.E-15. The scale factor S can be used to ensure this constraint while retaining the same units.

5. The output for the element is specified by the STRESS Case Control command and consists of the resistance, reactance, and absorption coefficient. The absorption coefficient is defined as:

$$a = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

**PACABS****Acoustic Absorber Property**

Defines the properties of the acoustic absorber element.

**Format:**

| 1      | 2   | 3     | 4    | 5    | 6    | 7 | 8     | 9 | 10 |
|--------|-----|-------|------|------|------|---|-------|---|----|
| PACABS | PID | SYNTH | TID1 | TID2 | TID3 |   | CUTFR | B |    |
|        | K   | M     |      |      |      |   |       |   |    |

**Example:**

|        |    |  |   |   |   |     |       |  |  |
|--------|----|--|---|---|---|-----|-------|--|--|
| PACABS | 12 |  | 1 | 2 | 3 | 3.5 | 500.0 |  |  |
|--------|----|--|---|---|---|-----|-------|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| SYNTH      | Request the calculation of B, K, and M from the tables TIDi below. (Character = “YES” or “NO”; Default = “YES”) |
| TID1       | Identification of the TABLEDi entry that defines the resistance. See Remark 2. (Integer > 0 or blank)           |
| TID2       | Identification of the TABLEDi entry that defines the reactance. See Remark 2. (Integer > 0 or blank)            |
| TID3       | Identification of the TABLEDi entry that defines the weighting function. See Remark 2. (Integer > 0 or blank)   |
| CUTFR      | Cutoff frequency for tables referenced above. (Real > 0.0)  |
| B, K, M    | Equivalent damping, stiffness and mass values per unit area. (Real $\geq 0.0$ )                                 |

**Remarks:**

1. PACABS is referenced by a CHACAB entry only.
2. If SYNTH = “YES”, then TID1 and TID2 must be supplied (TID3 is optional) and the equivalent structural model will be derived from tables TIDi. If TID3 is blank, then the weighting function defaults to 1.0.
3. If SYNTH = “NO”, then the equivalent structural model will be derived from one of B, K, or M.
4. The continuation entry is optional.
5. All data defined in tables TIDi must be a function of frequency in cycles/unit time.

**PACBAR**

## Acoustic Barrier Property

Defines the properties of the acoustic barrier element.

**Format:**

| 1      | 2   | 3     | 4      | 5      | 6      | 7 | 8 | 9 | 10 |
|--------|-----|-------|--------|--------|--------|---|---|---|----|
| PACBAR | PID | MBACK | MSEPTM | FRESON | KRESOM |   |   |   |    |

**Example:**

|        |    |     |      |       |  |  |  |  |  |
|--------|----|-----|------|-------|--|--|--|--|--|
| PACBAR | 12 | 1.0 | 0.01 | 400.0 |  |  |  |  |  |
|--------|----|-----|------|-------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)                                   |
| MBACK      | Mass per unit area of the backing material. (Real > 0.0)                        |
| MSEPTM     | Mass per unit area of the septum material. (Real > 0.0)                         |
| FRESON     | Resonant frequency of the sandwich construction in hertz. (Real > 0.0 or blank) |
| KRESOM     | Resonant stiffness of the sandwich construction. (Real > 0.0 or blank)          |

**Remarks:**

1. PACBAR is referenced by a CHACBR entry only.
2. Either FRESON or KRESOM must be specified, but not both.

**PACINF****Acoustic Conjugate Infinite Element Property**

Defines the properties of acoustic conjugate infinite elements.

**Format:**

| 1      | 2   | 3   | 4   | 5  | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|-----|----|----|----|---|---|----|
| PACINF | PID | MID | RIO | XP | YP | ZP |   |   |    |

**Example:**

|        |     |    |   |    |    |    |  |  |  |
|--------|-----|----|---|----|----|----|--|--|--|
| PACINF | 100 | 10 | 5 | 0. | 1. | 2. |  |  |  |
|--------|-----|----|---|----|----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property Identification Number of PACINF entry. (Integer > 0)                      |
| MID        | Material Identification Number of a MAT10 entry. (Integer > 0)                     |
| RIO        | Radial Interpolation Order. (Integer > 0; no Default)                              |
| XP, YP, ZP | Coordinates of the Pole of the Infinite Elements (in the Basic Coordinate System). |

**Remark:**

1. The location of the pole together with the connecting grid points of the element, define the geometry of the element, see [Figure 9-116](#).

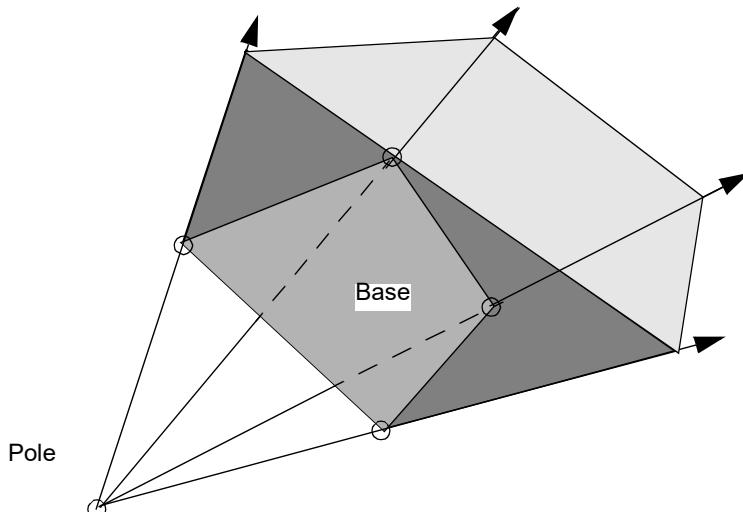


Figure 9-116 Geometry of Infinite Element

2. The radial interpolation order required depends on the directivity of the pressure field.

**PAERO1****Aerodynamic Panel Property**

Defines associated bodies for the panels in the Doublet-Lattice method.

**Format:**

| 1      | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|-----|----|----|----|----|----|----|---|----|
| PAERO1 | PID | B1 | B2 | B3 | B4 | B5 | B6 |   |    |

**Example:**

|        |   |   |  |  |  |  |  |  |  |
|--------|---|---|--|--|--|--|--|--|--|
| PAERO1 | 1 | 3 |  |  |  |  |  |  |  |
|--------|---|---|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number referenced by a CAERO1 entry. (Integer > 0)  |
| Bi         | Identification number of CAERO2 entries for associated bodies. Embedded blanks are not allowed. (Integer $\geq 0$ or blank) |

**Remarks:**

1. The associated bodies must be in the same aerodynamic group, as specified in the IGID field on CAERO2 entry.
2. If there are no bodies, the entry is still required (with Bi fields blank).
3. The Bi numbers above must appear on a CAERO2 entry to define these bodies completely.

**PAERO2****Aerodynamic Body Properties**

Defines the cross-sectional properties of aerodynamic bodies.

**Format:**

| 1      | 2    | 3      | 4     | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|------|--------|-------|------|------|------|------|------|----|
| PAERO2 | PID  | ORIENT | WIDTH | AR   | LRSB | LRIB | LTH1 | LTH2 |    |
|        | THI1 | THN1   | THI2  | THN2 | THI3 | THN3 |      |      |    |

**Example:**

|        |   |   |     |     |    |    |     |  |  |
|--------|---|---|-----|-----|----|----|-----|--|--|
| PAERO2 | 2 | Z | 6.0 | 1.0 | 22 | 91 | 100 |  |  |
|        | 1 | 3 |     |     |    |    |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0)  |
| ORIENT     | Orientation flag. Type of motion allowed for bodies. Refers to the aerodynamic coordinate system of ACSID. See AERO entry. (Character = "Z", "Y", or "ZY")   |
| WIDTH      | Reference half-width of body and the half-width of the constant width interference tube. (Real > 0.0)  |
| AR         | Aspect ratio of the interference tube (height/width). (Real > 0.0)   |
| LRSB       | Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the slender body elements. If blank or zero, the value of WIDTH will be used. (Integer $\geq 0$ or blank)      |
| LRIB       | Identification number of an AEFACT entry containing a list of interference body half-widths at the end points of the interference elements. If blank or zero, the value of WIDTH will be used. (Integer $\geq 0$ or blank) |
| LTH1, LTH2 | Identification number of AEFACT entries for defining $\theta$ arrays for interference calculations. (Integer $\geq 0$ )  |
| THIi, THNi | The first and last interference element of a body to use the $\theta_1$ array; the others use the $\theta_2$ array. (Integer $\geq 0$ )  |

**Remarks:**

1. The half-widths (given on AEFACT entries referenced in fields 6 and 7) are specified at division points. The number of entries on an AEFACT entry used to specify half-widths must be one greater than the number of elements.
2. The half-width at the first point (i.e., the nose) on a slender body is usually 0.0; thus, it is recommended (but not required) that the LRSB data is supplied with a zero first value.
3. THIi and THNi are interference element numbers on a body. The first element is one for each body.

4. A body is represented by a slender body surrounded by an interference tube. The slender body creates the downwash due to the motion of the body, while the interference tube represents the effects upon panels and other bodies.

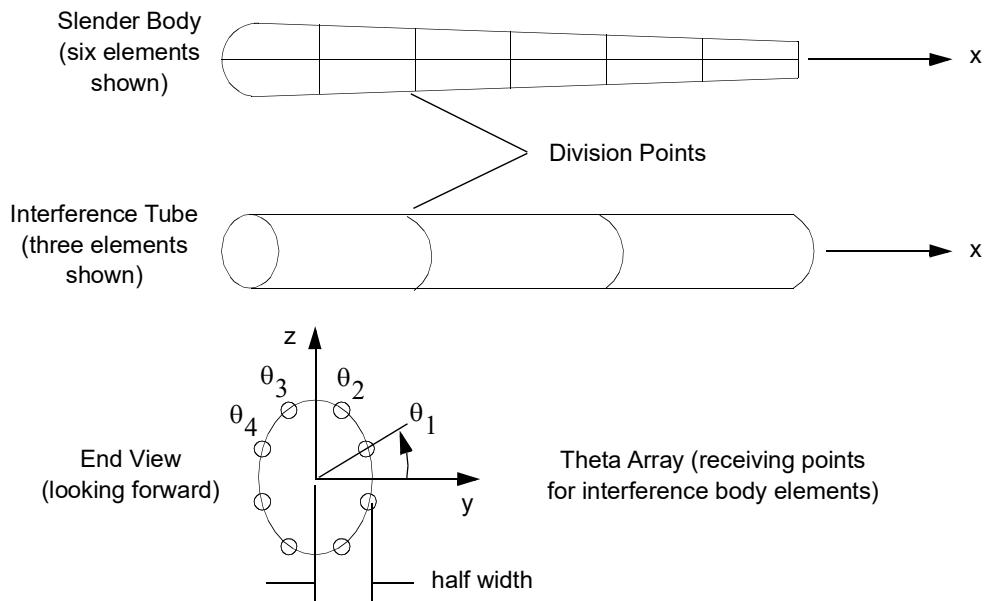


Figure 9-117 Idealization of Aerodynamic Body

5. The angles  $\theta_1$  and  $\theta_2$  are input in degrees using the aerodynamic element coordinate system as the reference coordinate system for defining the theta points.
6. Distribution of the theta points need not be uniform. A theta point must be placed a finite distance from any aerodynamic box edge; preferably the box edge would be equidistant from any two theta points. This aerodynamic coordinate system is defined on the AERO Bulk Data entry.
7. For half models, the theta arrays LTH1 and LTH2 should encompass a full 360 degree range.

**PAERO3****Aerodynamic Panel Property**

Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.

**Format:**

| 1      | 2   | 3    | 4     | 5   | 6  | 7  | 8   | 9   | 10 |
|--------|-----|------|-------|-----|----|----|-----|-----|----|
| PAERO3 | PID | NBOX | NCTRL |     | X5 | Y5 | X6  | Y6  |    |
|        | X7  | Y7   | X8    | Y8  | X9 | Y9 | X10 | Y10 |    |
|        | X11 | Y11  | X12   | Y12 |    |    |     |     |    |

**Example:**

|        |      |      |      |      |     |      |      |      |  |
|--------|------|------|------|------|-----|------|------|------|--|
| PAERO3 | 2001 | 15   | 1    |      | 0.  | 65.  |      |      |  |
|        | 78.  | 65.  | 108. | 65.  | 82. | 97.5 | 112. | 97.5 |  |
|        | 86.  | 130. | 116. | 130. |     |      |      |      |  |

| Descriptor     | Meaning   |
|----------------|---|
| PID            | Property identification number. (Integer > 0)   |
| NBOX           | Number of Mach boxes in the flow direction. (0 < Integer < 50)  |
| NCTRL          | Number of control surfaces. (Integer 0, 1, or 2)  |
| X5 through Y12 | Locations of points 5 through 12, which are in the aerodynamic coordinate system, to define the cranks and control surface geometry. (Real) |

**Remarks:**

- For an illustration of the geometry, see the CAERO3 entry description.
- If  $Y5 \leq 0.0$ , there is no leading edge crank. Also, if  $Y6 \leq 0.0$ , there is no trailing edge crank.
- If  $NCTRL = 0$ , no continuations are required. If  $NCTRL = 1$  or  $2$ , then NCTRL continuations are required.
- $Y7 \geq Y8, Y9 \geq Y10$ , and  $Y11 \geq Y12$ .
- The number of Mach boxes in the spanwise direction (NSB) may be found from the following formula:

$$NSB = \text{INT} \left[ \frac{\beta \cdot y_{max}}{\left( \frac{x_{max}}{NBOX + 0.5} \right)} + 0.5 \right]$$

where:

$$\beta = \sqrt{M^2 - 1}$$

$x_{max}$  = maximum chordwise direction

$y_{max}$  = maximum spanwise direction

NBOX = initial number of boxes specified in field 3

The number of Mach boxes in the streamwise direction may then be computed from:

$$\text{NBOX} = \text{INT} \left[ \frac{x_{max}}{\left( \frac{\beta \cdot y_{max}}{NSB - 0.5} \right)} + 0.999 \right]$$

The number of chordwise boxes specified by the user ( $\text{NBOX} \geq 50$ ) will be replaced by a floating point number (usually slightly higher than NBOX). The method contracts the mesh equally in both dimensions until a box edge lies on the surface tip. This mesh size is then used to compute the number of chordwise boxes.

**Note:**

A minimum of seven Mach boxes in the flow direction (NBOX) is recommended.

**PAERO4****Aerodynamic Strip Properties**

Defines properties of each strip element for Strip theory.

**Format:**

| 1      | 2    | 3     | 4      | 5    | 6     | 7      | 8      | 9      | 10 |
|--------|------|-------|--------|------|-------|--------|--------|--------|----|
| PAERO4 | PID  | CLA   | LCLA   | CIRC | LCIRC | DOC1   | CAOC1  | GAPOC1 |    |
|        | DOC2 | CAOC2 | GAPOC2 | DOC3 | CAOC3 | GAPOC3 | -etc.- |        |    |

**Example:**

|        |      |      |      |      |      |     |     |     |  |
|--------|------|------|------|------|------|-----|-----|-----|--|
| PAERO4 | 6001 | 1    | 501  | 0    | 0    | 0.0 | 0.0 | 0.0 |  |
|        | 0.50 | 0.25 | 0.02 | 0.53 | 0.24 | 0.0 |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| CLA        | Select Prandtl-Glauert correction. (Integer = -1, 0, 1; Default = 0) <ul style="list-style-type: none"> <li>-1 Compressibility correction made to lift curve slope data for a reference Mach number.</li> <li>0 No correction and no list needed. (Default)</li> <li>+1 No correction and lift curve slope provided by a list as a function of strip location and Mach number.</li> </ul> |
| LCLA       | Identification number of the AEFFACT entry that lists the lift curve slope on all strips for each Mach number on the MKAEROi entry. See Remark 7(b.) below. (Integer = 0 if CLA = 0, > 0 if CLA ≠ 0)  |
| CIRC       | Select Theodorsen's function $C(k)$ or the number of exponential coefficients used to approximate $C(k)$ . (Integer = 0, 1, 2, 3; Default = 0; Must be zero if CLA ≠ 0.) <ul style="list-style-type: none"> <li>0 Theodorsen function.</li> <li>1, 2, 3 Approximate function with <math>b_0, b_1, \beta_1, \dots, b_n, \beta_n</math> <math>n = 1, 2, 3</math>.</li> </ul>                |
| LCIRC      | Identification number of the AEFFACT entry that lists the $b, \beta$ values for each Mach number. See Remark 7c., 7d., and 7e. below; variable $b$ 's and $\beta$ 's for each mi on the MKAEROi entry. (Integer = 0 if CIRC = 0, > 0 if CIRC ≠ 0)   |
| DOCi       | $d/c$ distance of the control surface hinge aft of the quarter-chord divided by the strip chord. (Real $\geq 0.0$ )   |
| CAOCi      | $c_a/c$ control surface chord divided by the strip chord. (Real $\geq 0.0$ )  |
| GAPOCi     | $g/c$ control surface gap divided by the strip chord. (Real $\geq 0.0$ )  |

**Remarks:**

- PAERO4 is required for Strip theory with three fields (DOCi, CAOCi, GAPOCi) specified per strip.

2. If CLA = -1, lift curve slope data at one Mach number are needed on the AEFACT entry.
3. If CAOCi = 0.0, there is no control surface.
4. If GAPOCivb = 0.0, there is no slot flow.
5. If GAPOCi < 0.01, then 0.01 is used.
6. Embedded blank fields are not permitted.
7. [Table 9-28](#) lists the lift curve slope or lag function selection and the AEFACT entry formats used for Strip theory:

Table 9-28 Strip Theory Function Selections and AEFACT Entry Formats

| Theodorsen Function      | Data Type Input  | Parameter Combinations |      |      |       | Number of Words          | Entry Format Index |
|--------------------------|--|------------------------|------|------|-------|--------------------------|--------------------|
|                          |  | CLA                    | LCLA | CIRC | LCIRC |                          |                    |
| Exact                    | Lift Curve Slope<br>$c_{l\alpha_i} = 2\pi$             | 0                      | 0    | 0    | 0     | No AEFACT entry required |                    |
|                          | $c_{l\alpha_i}$ Input, Uses Prandtl-Glauert Correction | -1                     | ID   | 0    | 0     | (NSTRIP+1)               | a.                 |
|                          | $c_{l\alpha_i}$ Input, for All m's on MKAERO Entry     | 1                      | ID   | 0    | 0     | (NSTRIP+1)*NMACH         | b.                 |
| Approximate Coefficients | $b_{0i}, b_{1i}, \dots, b_{li}$ , etc.                 | 0                      | 0    | 1    | ID    | 4*NMACH                  | c.                 |
|                          |  | 0                      | 0    | 2    | ID    | 6*NMACH                  | d.                 |
|                          |  | 0                      | 0    | 3    | ID    | 8*NMACH                  | e.                 |

## Entry Format

- a. AEFACT, ID,  $m_1, c_{l\alpha_1}, c_{l\alpha_2}, \dots, c_{l\alpha_{NSTRIP}}$
- b. AEFACT, ID,  $m_1, c_{l\alpha_{11}}, c_{l\alpha_{21}}, \dots, c_{l\alpha_{NSTRIP1}}, m_2, c_{l\alpha_{11}}, c_{l\alpha_{12}}, c_{l\alpha_{21}}, c_{l\alpha_{22}}, \dots, c_{l\alpha_{NSTRIP1}}, c_{l\alpha_{NSTRIP2}}$ , for all m on MKAEROi data entry
- c. AEFACT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, m_2, b_{02}, b_{12}, P_{12}, m_3$ , etc.
- d. AEFACT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, m_2$ , etc.
- e. AEFACT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, b_{31}, \beta_{31}, m_2$  etc.
8. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.

**PAERO5****Aerodynamic Panel Property**

Defines properties of each strip element for Piston theory.

**Format:**

| 1      | 2     | 3      | 4      | 5     | 6     | 7     | 8     | 9 | 10 |
|--------|-------|--------|--------|-------|-------|-------|-------|---|----|
| PAERO5 | PID   | NALPHA | LALPHA | NXIS  | LXIS  | NTAUS | LTAUS |   |    |
|        | CAOC1 | CAOC2  | CAOC3  | CAOC4 | CAOC5 |       |       |   |    |

**Example:**

|        |      |     |      |         |     |   |     |  |  |
|--------|------|-----|------|---------|-----|---|-----|--|--|
| PAERO5 | 7001 | 1   | 702  | 1       | 701 | 1 | 700 |  |  |
|        | 0.0  | 0.0 | 5.25 | 3.99375 | 0.0 |   |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Unique Integer > 0)   |
| NALPHA     | Number of angle of attack ( $\alpha$ ) values to be input for each Mach number (mi) on the MKAERO1 or MKAERO2 entry. (Integer > 0) |

| NALPHA           | Meaning   |
|------------------|---|
| 1                | $\alpha$ is the same value for all strips; enter one value, in units of degrees, on the AEFACT entry for each Mach number.  |
| Number of Strips | $\alpha$ is different for each strip; enter $\alpha$ 's, in units of degrees, in the following order: $m_1, \alpha_1, \alpha_2, \dots, m_2, \alpha_1, \alpha_2, \dots$ , etc. |

|        |  |
|--------|--|
| LALPHA | ID number of the AEFACT entry that lists the $\alpha$ 's for the strips at each Mach number in the MKAERO1 or MKAERO2 entry. (Integer > 0) |
| NXIS   | Number of dimensionless chord coordinates ( $\xi$ ) to be input. (Integer $\geq 0$ , Default = 0)  |

| NXIS             | Meaning  |
|------------------|--|
| 0                | No $\xi$ 's are required. (Default)  |
| 1                | $\xi$ 's are the same for all strips; enter values for one strip on the AEFACT entry ( $\xi_h$ if NTHICK > 0, or $\xi_m$ and $\xi_h$ if NTHICK = 0)  |
| Number of Strips | $\xi$ 's have to be input for each strip ( $\xi_{h1}, \xi_{h2}, \dots, \xi_{hNSPAN}$ , if NTHICK > 0, or $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSPAN}, \xi_{hNSPAN}$ , $\beta_{hNSPAN}$ if NTHICK = 0) |

|       |   |
|-------|---|
| LXIS  | Identification number of AEFACT entry that lists the $\xi$ values for the strip in order indicated by values of NXIS and NTHICK. (Integer = 0 if $c_a = 0$ and NTHICK > 0 or LXIS>0 if $c_a = 0$ and/or NTHICK = 0) |
| NTAUS | Parameter used to select the number of thickness ratio ( $\tau$ ) values to be input. (Integer $\geq 0$ , Default = 0)  |

| NTAUS            | Meaning  |
|------------------|--|
| 0                | No $\tau$ 's are required. (Default)   |
| 1                | $\tau$ 's are the same for all strips; enter $(\tau_1, \tau_{h1}, \tau_{t1})$ values for one strip on AEFACT entry.  |
| Number of Strips | $\tau$ 's must be input for each strip on an AEFACT entry in the following order:<br><br>$(\tau_1, \tau_{h1}, \tau_{t1}, \tau_2, \tau_{h2}, \tau_{t2}, \dots, \tau_{NSPAN}, \tau_{hNSPAN}, \tau_{tNSPAN})$ |

|       |  |
|-------|--|
| LTAUS | Identification number of AEFACT entry that lists the $\tau$ values for the strips. (Integer = 0 or blank if NTAUS = 0, LTAUS > 0 if NTAUS > 0) |
| CAOCi | $c_a/c$ = control surface chord divided by the strip chord. (Real $\geq 0.0$ )   |

### Remarks:

1. The continuation entry is required for Piston theory with one entry (CAOCi) per strip.
2. Embedded blank fields are not allowed on the continuation entry.
3. If CAOCi = 0.0, there is no control surface.
4. [Table 9-29](#) lists the thickness data input and AEFACT entry format used for Piston theory.

Table 9-29 Thickness Data Input and AEFACT Entry Format for Piston Theory

| Type of Input   | Parameter Combinations |                   |        |                   |        |                   | Number of Words      | Entry Format Index |
|---|------------------------|-------------------|--------|-------------------|--------|-------------------|----------------------|--------------------|
|   | CAOG                   | NGHICK            | NXIS   | LXIS              | NTAUS  | LTAUS             |                      |                    |
| No control surfaces,<br>Integrals input are same for all strips     | 0.                     | ID <sup>(a)</sup> | 0      | 0                 | 0      | 0                 | 6                    | a.                 |
| With control surfaces,<br>Integrals input, same hinge on all strips | $\neq 0$               | ID <sup>(b)</sup> | 1      | ID <sup>(c)</sup> | 0      | 0                 | 12<br>1              | b.<br>c.           |
| With control surfaces,<br>Integrals input, variable hinge           | $\neq 0$               | ID <sup>(b)</sup> | NSTRIP | ID <sup>(d)</sup> | 0      | 0                 | 12<br>NSTRIP         | b.<br>d.           |
| No control surfaces, thickness inputs are same for all strips       | 0.0                    | 0                 | 1      | ID <sup>(f)</sup> | 1      | ID <sup>(e)</sup> | 3<br>2               | e.<br>f.           |
| With control surfaces, thickness inputs are same for all strips     | $\neq 0.0$             | 0                 | 1      | ID <sup>(f)</sup> | 1      | ID <sup>(e)</sup> | 3<br>2               | e.<br>f.           |
| With control surfaces, thickness inputs vary for strips             | $\neq 0.0$             | 0                 | NSTRIP | ID <sup>(h)</sup> | NSTRIP | ID <sup>(g)</sup> | 3*NSTRIP<br>2*NSTRIP | g.<br>h.           |

## Entry Format

- a. AEFACT, ID,  $I_1, I_2, I_3, I_4, I_5, I_6$
- b. AEFACT, ID,  $I_1, \dots, I_6, J_1, \dots, J_6, I_1, I_2, I_3, I_4, I_5, I_6$
- c. AEFACT, ID,  $\xi_h$
- d. AEFACT, ID,  $\xi_{h1}, \xi_{h2}, \xi_{h3}, \dots, \xi_{hNSTRIP}$
- e. AEFACT, ID,  $\tau_m, \tau_h, \tau_t$
- f. AEFACT, ID,  $\xi_m, \xi_h$
- g. AEFACT, ID,  $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$   
 $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$
- h. AEFACT, ID,  $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSTRIP}, \xi_{hNSTRIP}$

5. The following table lists the angle-of-attack distribution and AEFACT entry formats used for Piston theory.

| Type of Distribution                | Parameter Combinations |        | Number of Words      | Entry Format Index |
|-------------------------------------|------------------------|--------|----------------------|--------------------|
|                                     | NALPHA                 | LALPHA |                      |                    |
| Equal angle of attack on all strips | 1                      | ID     | 2*NMACH              | a.                 |
| Unequal angle of attack             | NSTRIP                 | ID     | (1 + NSTRIP) * NMACH | b.                 |

#### Entry Format

- a. AEFACT, ID,  $m_1, \alpha_1, m_2, \alpha_2, \dots,$
- b. AEFACT, ID,  $m_1, \alpha_{11}, \alpha_{21}, \alpha_{31}, \dots, \alpha_{NSTRIP1}, m_2, \alpha_{12}, \alpha_{22}, \dots, \alpha_{NSTRIP2}, m_2,$  etc., for all m on MKAEROi entry.
- c. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.

**PANEL****Panel Definition for Coupled Fluid-Structural Analysis**

Defines one or more panels by referencing sets of grid points, elements or properties.

**Format:**

| 1     | 2     | 3      | 4     | 5      | 6     | 7      | 8     | 9      | 10 |
|-------|-------|--------|-------|--------|-------|--------|-------|--------|----|
| PANEL | NAME1 | SETID1 | NAME2 | SETID2 | NAME3 | SETID3 | NAME4 | SETID4 |    |

**Example:**

|       |        |     |  |  |  |  |  |  |  |
|-------|--------|-----|--|--|--|--|--|--|--|
| PANEL | BKDOOR | 103 |  |  |  |  |  |  |  |
|-------|--------|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| NAMEi      | Panel label. (Character)  |
| SETIDi     | Identification number of a SET1 or SET3 entry that lists the grid points, elements or properties of the panel. On this entry, the SET1 should only be used for grid definition. (Integer > 0) |

**Remarks:**

1. If a set of grid points is referenced, the set must list only structural grid points.
2. If an element is assigned to a panel, it is recommended that all of its connection points belong to the same panel.
3. If a set of elements is referenced, the set must list only structural elements. The panel will consist of all grid points that are connection points of these elements.
4. If a set of property identifiers is referenced, the properties must be referenced by structural elements. The panel will consist of all grid points that are connection points of elements referencing one of the properties contained in the set.
5. NAMEi is used only for labeling the output of the panel participation factors (cf. the description of the PFMODE and PFPANEL Case Control commands).

**PARAM**

## Parameter

Specifies values for parameters used in solution sequences or user-written DMAP programs.

**Format:**

| 1     | 2 | 3  | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|---|----|----|---|---|---|---|---|----|
| PARAM | N | V1 | V2 |   |   |   |   |   |    |

**Example:**

|       |      |   |  |  |  |  |  |  |  |
|-------|------|---|--|--|--|--|--|--|--|
| PARAM | IRES | 1 |  |  |  |  |  |  |  |
|-------|------|---|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| N          | Parameter name (one to eight alphanumeric characters, the first of which is alphabetic). |
| V1, V2     | Parameter value based on parameter type, as follows:                                     |

| Type                      | V1                    | V2                    |
|---------------------------|-----------------------|-----------------------|
| Integer                   | Integer               | Blank                 |
| Real, single-precision    | Real                  | Blank                 |
| Character                 | Character             | Blank                 |
| Real, double-precision    | Double-precision real | Blank                 |
| Complex, single-precision | Real or blank         | Real or blank         |
| Complex, double-precision | Double-precision real | Double-precision real |

**Remarks:**

- See [Parameters, 783](#) for a list of parameters used in solution sequences that may be set by the user on PARAM entries.
- If the large field entry format is used, the second physical entry line must be present, even though fields 6 through 9 are blank except for SOL 700.
- The first 8 characters of N must be unique. N with more than 8 characters is normally only used by SOL 700.
- If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:
  - PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
  - PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.

- c. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.
- 5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**PARAMARC**

## Parallel Domain Decomposition in Marc from SOL 600

Specifies parallel regions for domain decomposition in nonlinear analysis when Marc is executed from MSC Nastran. Used in SOL 600 only.

**Format:**

| 1        | 2   | 3    | 4     | 5       | 6     | 7   | 8   | 9   | 10 |
|----------|-----|------|-------|---------|-------|-----|-----|-----|----|
| PARAMARC | ID  | KIND | NPROC | IDSOLVE | ACCUR |     |     |     |    |
|          | IDP | ID1  | ID2   | ID3     | ID4   | ID5 | ID6 | ID7 |    |
|          | IDP | ID8  | ID9   | etc.    |       |     |     |     |    |

**Example: To create 4 parallel processes using Marc's single file input**

|          |    |  |   |   |        |  |  |  |  |
|----------|----|--|---|---|--------|--|--|--|--|
| PARAMARC | 51 |  | 4 | 0 | 1.0E-4 |  |  |  |  |
|----------|----|--|---|---|--------|--|--|--|--|

**Example: To create 2 parallel processes by specifying element numbers**

|          |     |      |      |      |      |      |      |      |  |
|----------|-----|------|------|------|------|------|------|------|--|
| PARAMARC | 201 | 2    | 2    |      |      |      |      |      |  |
|          | 1   | 1    | THRU | 5000 | BY   | 1    |      |      |  |
|          | 2   | 5001 | 5002 | 5005 | 5010 | 5012 | 5013 | 5015 |  |
|          | 2   | 5020 | 5030 | 5040 | 5050 |      |      |      |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number of the PARAMARC entry -- Not presently used. (Integer)  |
| KIND       | Designates how parallel domains are created. (Integer = 0 or Blank; Default = 0)  |
| 0          | Parallel processing is accomplished using Marc's single file input. (Marc Version 2005 or subsequent must be used. The command line to execute Marc is changed from -np N (or -nproc N) to -nps N where N is the number of processors. The maximum number of processors for Marc is 256. Continuation lines may not be entered for KIND=0.) |
| 2          | Parallel domains will be specified by the user and the continuation lines are required.   |
| NPROC      | Number of parallel processes requested. See Remark 11. (Integer; Default = 0)   |
| IDSOLV     | Inter-domain solution type. See Remark 11. (Integer; Default = 0)   |
| 0          | Iterative Solver  |
| 1          | Direct Solver   |
| ACCUR      | Inter-domain solve accuracy. See Remark 11. (Real; Default = 1.0E-2) For an accurate solution of difficult nonlinear models, ACCUR should be set to 1.0E-4.   |

| Describer | Meaning  |
|-----------|--|
| IDP       | Parallel process ID for following group of elements or grid ID's. See Remark 5. (Integer > 0; Required)              |
| ID(i)     | Element number (if KIND=2). (Integer > 0; Required if KIND=2)  |
| THRU      | Enter THRU in field 4 (may be omitted if I2 and INC are not required. The word TO instead of THRU may also be used.) |
| ID2       | Ending element number. (Integer > 0)   |
| BY        | Enter BY in field 6 (may be omitted in INC=1)  |
| INC       | Increment ID's. (Integer; Default = 1, may be negative but not zero)   |

**Remarks:**

1. The PARAMARC entry is recognized only when Marc is executed from within SOL 600.
2. Use of KIND=2 provides more control than does the automatic (KIND=0) option.
3. If parallel jobs are run on different computers across a network, as opposed to using multiple processors in the same box, a host file is normally needed. Consult MSC technical support to determine how to setup a host file for your computer system. Use of the host file is triggered by Bulk Data PARAM,MARCHHOST,Name.
4. Continuation lines should not be entered unless KIND=2.
5. The continuation entries should be entered as many times as necessary to completely define each parallel region for KIND=2.
6. The string TO may be substituted for THRU if so desired.
7. For PC Windows systems, the default type of MPI for SOL 600 is Intel MPI. The first time a parallel job is run, the user may be prompted for domain\user\_name and then for password. Since MSC Nastran is a batch process, the user will not normally see the prompts and the job may appear to hang. If the job appears to hang, carefully enter the following information in exactly the same way you enter it to login into your PC:
 

```
domain\user_name
      password
```
8. If multiple computers are used across a network, all computers must normally be the same type of computer, run the same operating system, be in the same domain, have the same user name and passwords. Also, a host file is required to describe the machines to be used. Further details are provided in the *SOL 600 Parallel Processing User's Guide*.
9. See PARAM,MRPARALL for additional notes concerning SOL 600 parallel processing.
10. If running on a PC, see PARAM,MARCMPII for options to keep the small MPI service (for example, ismpd.exe for Intel MPI) running or not after the Marc portion of the job has completed.
11. Parallel processing for SOL 600 is accomplished using domain decomposition. A solution of each domain is performed using its own processor. When each of the individual processor solutions converge, the overall solution at each increment is put together from the domains and iterated until convergence occurs. Variables IDSOLV and ACCUR control how this final iteration is performed

and its accuracy. IDSOLV=0 (iterative solver) will usually be faster than IDSOLV=1 (direct solver), however it is sometimes necessary that ACCUR be reduced to 1.0E-3 or 1.0E-4 to obtain sufficient accuracy for difficult nonlinear models, particularly those with contact, post-buckling, or large strain plasticity.

**PAXISYM**

## Properties of Axisymmetric Line Element - SOL 400

Defines the properties of axisymmetric line elements.

**Format:**

| 1       | 2   | 3   | 4 | 5  | 6  | 7    | 8 | 9 | 10 |
|---------|-----|-----|---|----|----|------|---|---|----|
| PAXISYM | PID | MID |   | T1 | T2 | ANAL |   |   |    |

**Example:**

|         |    |    |  |     |  |  |  |  |  |
|---------|----|----|--|-----|--|--|--|--|--|
| PAXISYM | 98 | 17 |  | 0.1 |  |  |  |  |  |
|---------|----|----|--|-----|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number of a CAXISYM entry. (Integer > 0)  |
| MID        | Identification number of a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entry. Identification number of a MAT4 or MAT5 for heat transfer. See Remark 3. (Integer > 0) |
| T1         | Thickness at G1. (Real > 0)   |
| T2         | Thickness at G2. (Real $\geq$ 0)  |
| ANAL       | Analysis type, IS=structural, IH=heat, ISH=structural-heat (CHAR Default=ISH)   |

**Remarks:**

1. PAXISYM identification entries should be unique with respect to all other property entries.
2. For elements with only two grids, only T1 is applicable. For elements with three grids, if T2 is blank or 0.0 then constant thickness is assumed.
3. The MID entry may point to MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entries for structures and MAT4 or MAT5 entries for heat transfer. The tables below show associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

| Implicit Structural Materials |         |         |         |         |       |
|-------------------------------|---------|---------|---------|---------|-------|
| MAT1                          | MAT2    | MAT3    | MAT8    | MATORT  | MATHE |
| MATVE                         | <MATVE> | <MATVE> | <MATVE> | <MATVE> | MATVE |
| MATVP                         | MATVP   | MATVP   |         | MATVP   |       |
| MATEP                         | MATEP   | MATEP   | MATEP   | MATEP   |       |
| MATF                          | MATF    | MATF    | MATF    | MATF    |       |
| MATS1                         |         | MATS3   | MATS8   | MATSORT |       |

<MATVE> refers to the ALTERNATE format for type ORTHO

| Heat Materials |      |
|----------------|------|
| MAT4           |      |
|                | MAT5 |

MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries. MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The CAXISYM element uses PLOADX1 for loading.
5. The element does not support composite materials, via PCOMP or PCOMPG.

**PAXSYMH****Linear Axisymmetric Harmonic Element Properties**

Defines the properties of a linear axisymmetric harmonic element.

**Format:**

| 1       | 2   | 3   | 4   | 5     | 6   | 7 | 8 | 9 | 10 |
|---------|-----|-----|-----|-------|-----|---|---|---|----|
| PAXSYMH | PID | MID | CID | NHARM | INT |   |   |   |    |

**Example:**

|         |     |    |   |  |  |  |  |  |  |
|---------|-----|----|---|--|--|--|--|--|--|
| PAXSYMH | 100 | 10 | 5 |  |  |  |  |  |  |
|---------|-----|----|---|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0). See Remarks 1. and 5.                                 |
| MID        | Identification number of MAT1 or MAT9 entry. (Integer > 0).  |
| CID        | Identification number of element coordinate system. (Integer $\geq 0$ ; Default = 0). See Remark 2.. |
| NHARM      | Harmonic index. (Integer $\geq 0$ ; Default = 1). See Remark 3..                                     |
| INT        | Integration scheme. (Integer 0, 1, 2 or 3; Default = 0). See Remark 4.                               |

**Remarks:**

1. This entry is referenced by the CQUADX and CTRIA9 entries.
2. The Y-axis of CID defines the axis of symmetry while its X-Y plane defines the two-dimensional plane of the axisymmetric harmonic element. The X-axis of CID thus defines the radial direction.
3. The gyroscopic matrix is generated for the element only for the case of NHARM = 1 (the default).
4. The integration scheme INT selects the number of Gauss points used for matrix generation as indicated below:

| INT | Order | Gauss Points | Gauss Points |
|-----|-------|--------------|--------------|
|     |       | CQUADX       | CTRIAX       |
| 1   | 1x1   | 1            | 1            |
| 2   | 2x2   | 4            | 3            |
| 3   | 3x3   | 9            | 7            |

INT=0 uses INT=2 (no mid-side nodes) or INT=3 (mid-side nodes). The default INT=0 is recommended. The use of INT=1 will produce a singular stiffness and is meant for experimental use only.

5. PAXSYMH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PAXSYMH property entries have unique identification numbers with respect to all other property entries, else unexpected grouping results may occur. There must be uniqueness among PAXSYMH, PLPLANE and PSHELL entries.

**PBAR****Simple Beam Property**

Defines the properties of a simple beam element (CBAR entry).

**Format:**

| 1    | 2   | 3   | 4   | 5  | 6  | 7  | 8   | 9  | 10 |
|------|-----|-----|-----|----|----|----|-----|----|----|
| PBAR | PID | MID | A   | I1 | I2 | J  | NSM |    |    |
|      | C1  | C2  | D1  | D2 | E1 | E2 | F1  | F2 |    |
|      | K1  | K2  | I12 |    |    |    |     |    |    |

**Example:**

|      |    |   |     |     |      |  |  |  |  |
|------|----|---|-----|-----|------|--|--|--|--|
| PBAR | 39 | 6 | 2.9 |     | 5.97 |  |  |  |  |
|      |    |   |     | 2.0 | 4.0  |  |  |  |  |

| Descriptor     | Meaning  |
|----------------|--|
| PID            | Property identification number. (Integer > 0)  |
| MID            | Material identification number. See Remarks 2. and 3. (Integer > 0)  |
| A              | Area of bar cross section. (Real; Default = 0.0)   |
| I1, I2, I12    | Area moments of inertia. See <a href="#">Figure 9-118</a> . (Real; $I1 \geq 0.0$ , $I2 \geq 0.0$ , $I1*I2 \geq I12^2$ ; Default = 0.0)                 |
| J              | Torsional constant. See <a href="#">Figure 9-118</a> . (Real; Default = $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences) |
| NSM            | Nonstructural mass per unit length. (Real)   |
| Ci, Di, Ei, Fi | Stress recovery coefficients. (Real; Default = 0.0)  |
| K1, K2         | Area factor for shear. See Remark 5. (Real or blank)   |

**Remarks:**

- Both continuation entries may be omitted.
- For structural problems, MID must reference a MAT1 material entry.
- For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
- See the CBAR entry description for a discussion of bar element geometry.
- The transverse shear stiffnesses times unit length in planes 1 and 2 are  $K1*A*G$  and  $K2*A*G$ , respectively, where G is the shear modulus. The default values for K1 and K2 are infinite; in other words, the transverse shear flexibilities are set equal to zero. K1 and K2 are ignored if  $I12 \neq 0$ . K1 and K2 must be blank if A = 0.0.

6. The stress recovery coefficients C1 and C2, etc., are the y and z coordinates in the bar element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the bar. For conventional element, only bending components of strain and stress are outputted at points C1 and C2, etc. Membrane components of strain and stress are outputted as axial value. For advanced nonlinear element, the bending and membrane components of strain and stress are superposed and outputted together at the points C1 and C2, etc., there is no axial value individually.
7. For response spectra analysis on stress recovery coefficients, the CBEAM element entry should be used because bar element results will be inaccurate.
8. [Figure 9-118](#) describes the PBAR element coordinate system.

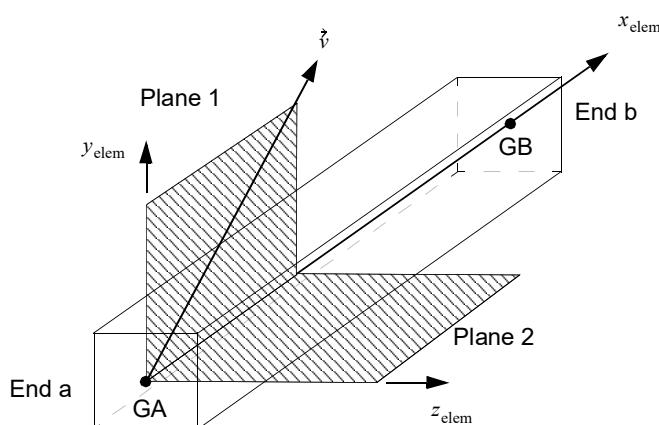
where:

$$I1 = I_{zz_{\text{elem}}}$$

$$I2 = I_{yy_{\text{elem}}}$$

$$I12 = I_{zy_{\text{elem}}}$$

$$J = I_{xx_{\text{elem}}}$$



[Figure 9-118](#) PBAR Element Coordinate System

9. For cross-sections that are not doubly symmetric, when a beam is loaded through the centroid, it may in addition to bending undergo rotation. The CBAR element, by default, does not represent this behavior because the shear center is not explicitly accounted for on a CBAR entry.

By definition, for the CBAR element, the load is applied at the centroid and not at the shear center. If load application at the shear center is desired, appropriately applied offsets can be used on the CBAR entry or the CBAR/PBAR element replaced by the CBEAM/PBEAM element which by default explicitly applies the load through the shear center.

If warping considerations and bi-moment calculations are important, the CBEAM/PBEAM element should be used.

10. Mass moment of inertial formulation has changed in Version 2003. System (398) may be used to select the formulation in pre-Version 2004 systems.
11. PBAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

**PBART**

## Simple Beam Cross-Section Property

Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.

**Format:**

| 1     | 2    | 3      | 4     | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|--------|-------|------|------|------|------|------|----|
| PBART | PID  | MID    | GROUP | TYPE |      |      |      |      |    |
|       | DIM1 | DIM2   | DIM3  | DIM4 | DIM5 | DIM6 | DIM7 | DIM8 |    |
|       | DIM9 | -etc.- | NSM   |      |      |      |      |      |    |

**Example:**

|       |     |    |    |    |    |    |    |  |  |
|-------|-----|----|----|----|----|----|----|--|--|
| PBART | 39  | 6  |    | I  |    |    |    |  |  |
|       | 14. | 6. | .5 | .5 | .5 | .5 | .2 |  |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| MID        | Material identification number. (Integer > 0)   |
| GROUP      | Cross-section group. See Remarks 6. and 8. (Character; Default = "MSCBMLO")   |
| TYPE       | Cross-section type. See Remarks 6. and 8. and Figure 9-119. (Character: "ROD", "TUBE", "TUBE2", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP="MSCBMLO") |
| DIMi       | Cross-sectional dimensions. (Real > 0.0 for GROUP = "MSCBMLO")  |
| NSM        | Nonstructural mass per unit length. NSM is specified after the last DIMi. (Default = 0.0)   |

**Remarks:**

- For structural problems, PBART entries must reference a MAT1 material entry.
- PBART is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBART property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBART, PBRSECT entries.
- See CBAR entry for a discussion of bar element geometry.
- For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
- For response spectra analysis on stress recovery coefficients, the CBEAM element should be used because results for the CBAR element will not be accurate.

6. The GROUP is associated with an FMS CONNECT statement that specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require one or more subroutines to convert DIMi information to geometric property information contained on a PBAR entry and optimization information. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion on how to include a user-defined beam library.
7. A function of this entry is to derive an equivalent PBAR entry. Any sorted echo request will also cause printout and/or punch of the derived PBAR.
8. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are computed using the TYPE and DIMi as shown in [Figure 9-119](#) through [Figure 9-122](#). The figures show the origin of the cross section, but the PBARL does not account for differences between the shear center and the neutral axis and the properties are computed relative to the neutral axis. This results in approximations for section types ‘T’, ‘CHAN’, ‘T’, ‘CHAN1’, ‘T1’, ‘CHAN2’, ‘T2’, ‘L’ and ‘BOXI’. The PBEAML provides a better representation of section properties in these cases and is recommended.

The PBARL does not account for offsets between the neutral axis and the shear center. Therefore, the CHAN, CHAN1 and CHAN2 cross-sections may produce incorrect results. The PBEAML is recommended.

9. For DBOX section, the default value for DIM5 to DIM10 are based on the following rules:
  - a. DIM5, DIM6, DIM7 and DIM8 have a default value of DIM4 if not provided.
  - b. DIM9 and DIM10 have a default value of DIM6 if not provided.

**Note:**

The above default value rules for DIM5 to DIM10 are not applicable to design optimization property value update.

10. The finite element formulation (FEF) utilized for arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBARL when GROUP=MSCBML0. The original beam equations which are based on thin-walled assumptions can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’. Thus the shear stiffness factors K1 and K2 are calculated as in Remark 11. of the PBMSECT entry and the element will behave according to the Timoshenko beam theory by default. Thus the resulting CBAR will have non-infinite K1/K2 shear flexibility factors, and therefore will not behave as a Bernoulli-Euler.
11. For optimization, individual DIMx of PBARL can be selected as designed properties even with the finite element formulation.

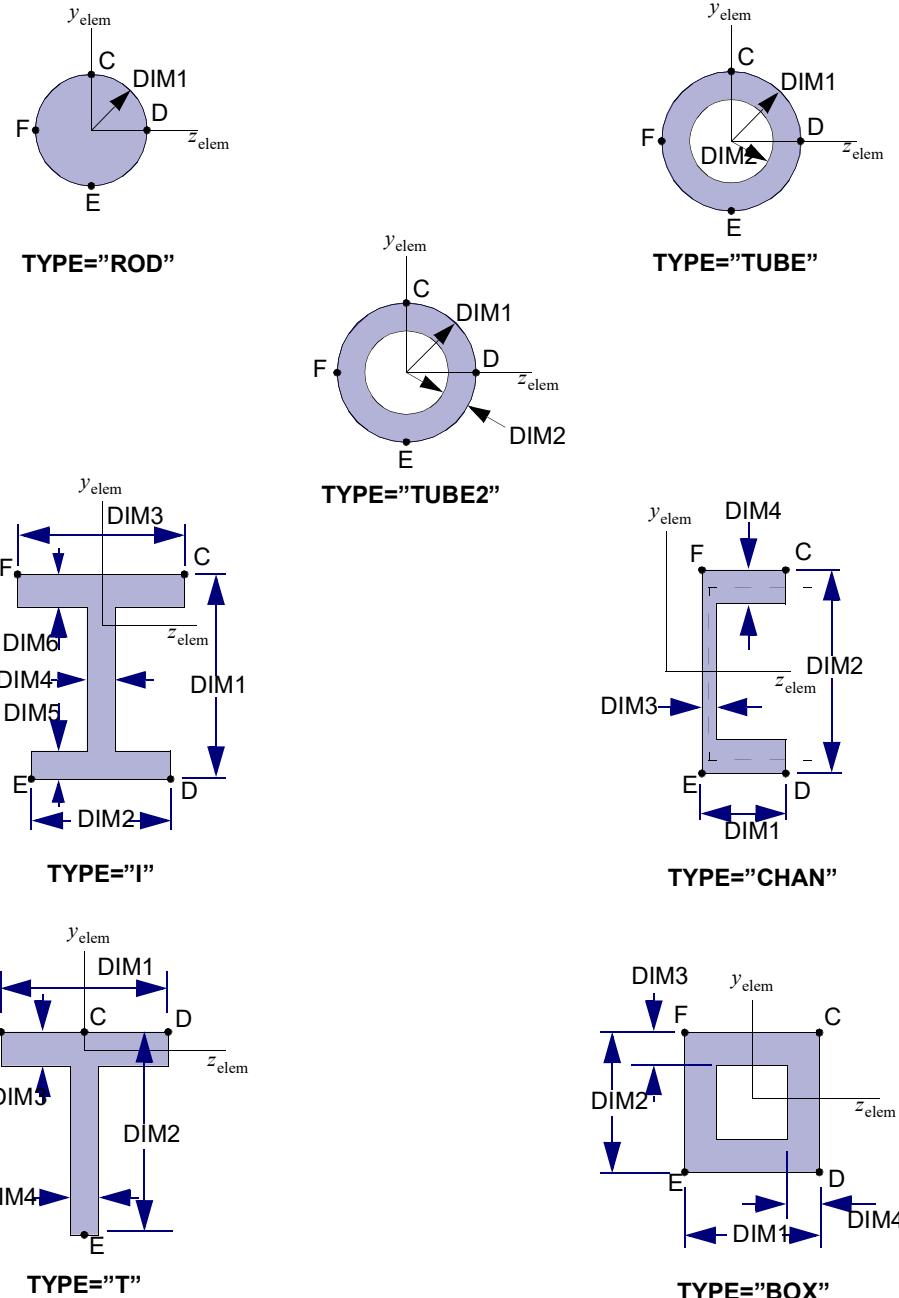


Figure 9-119 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

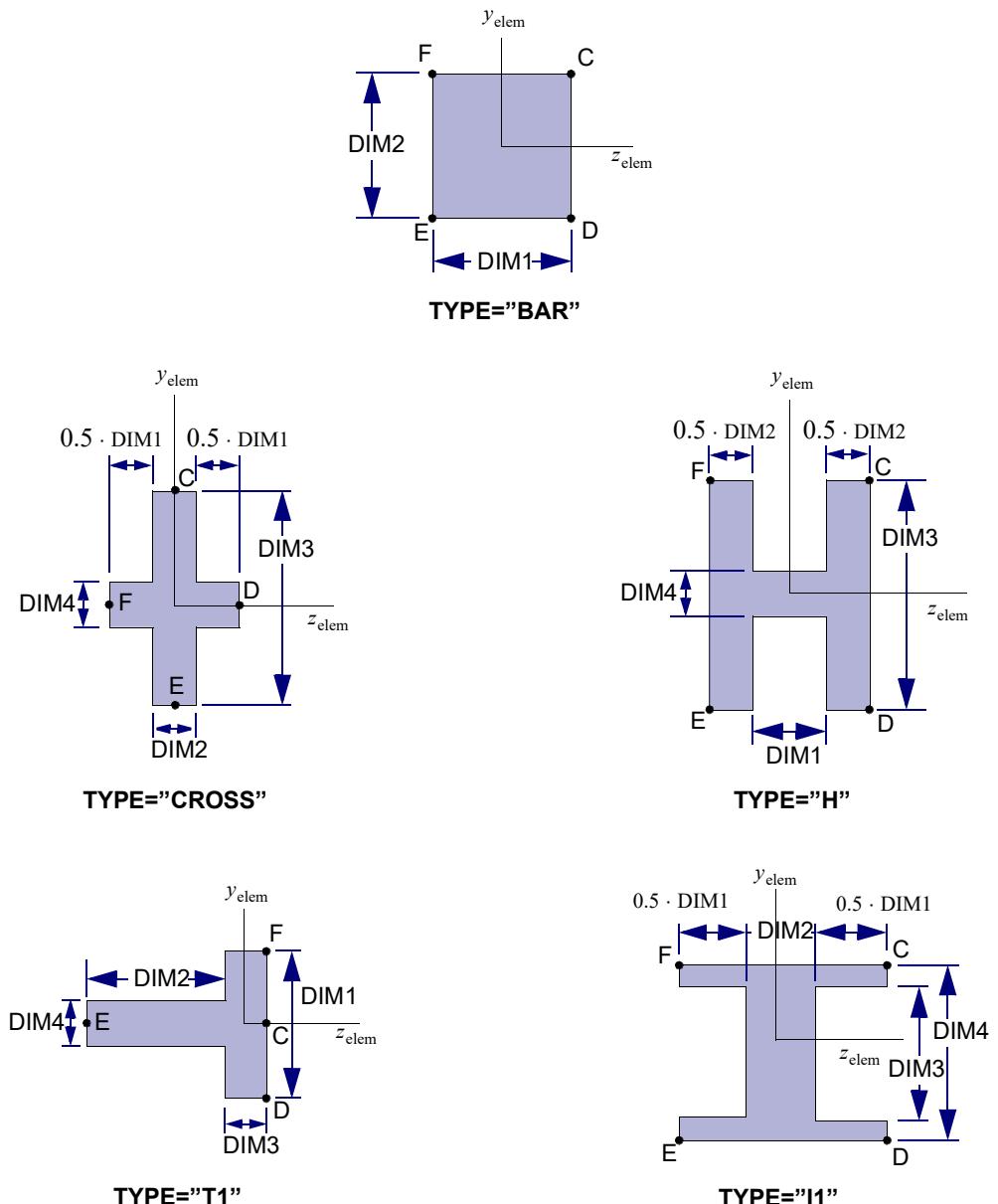


Figure 9-120 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

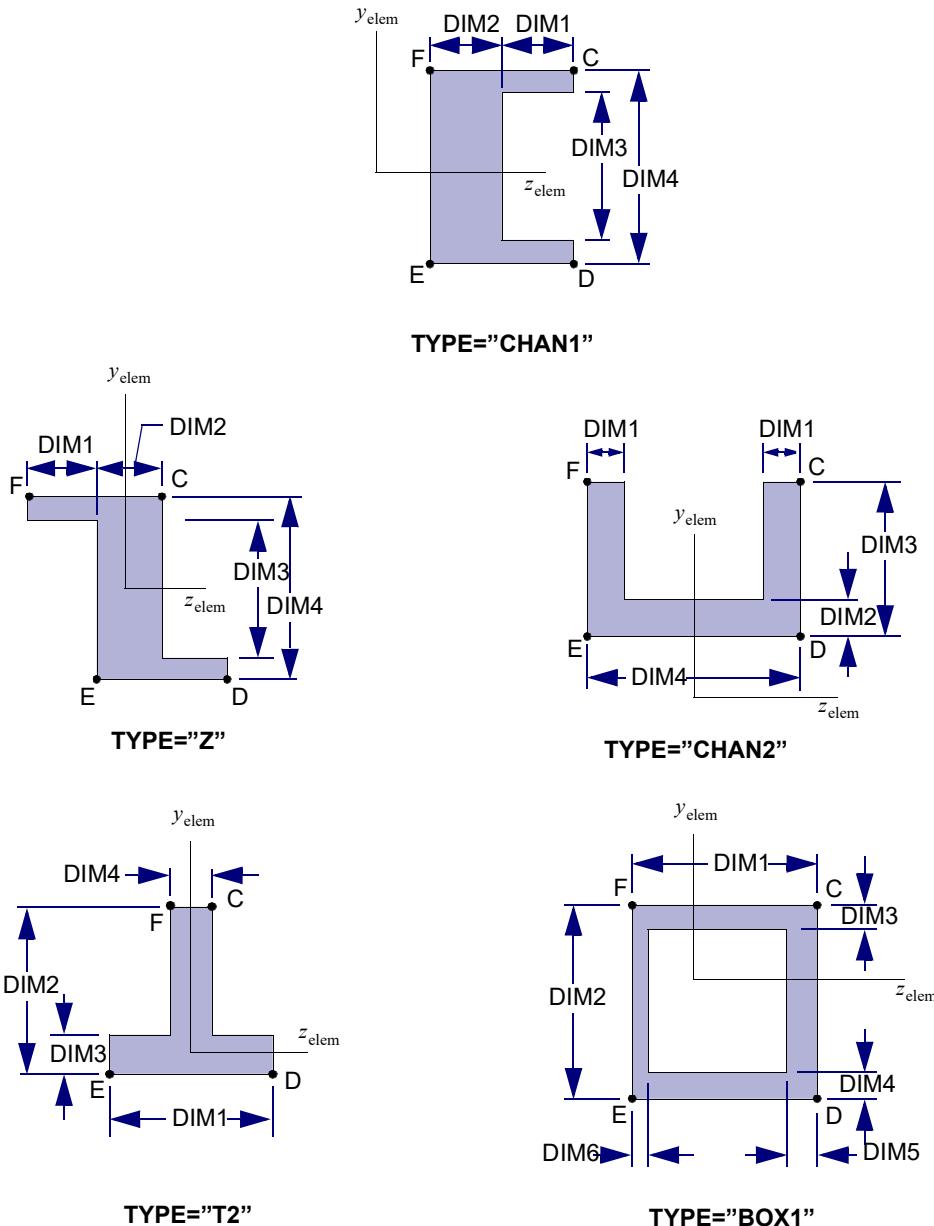


Figure 9-121 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

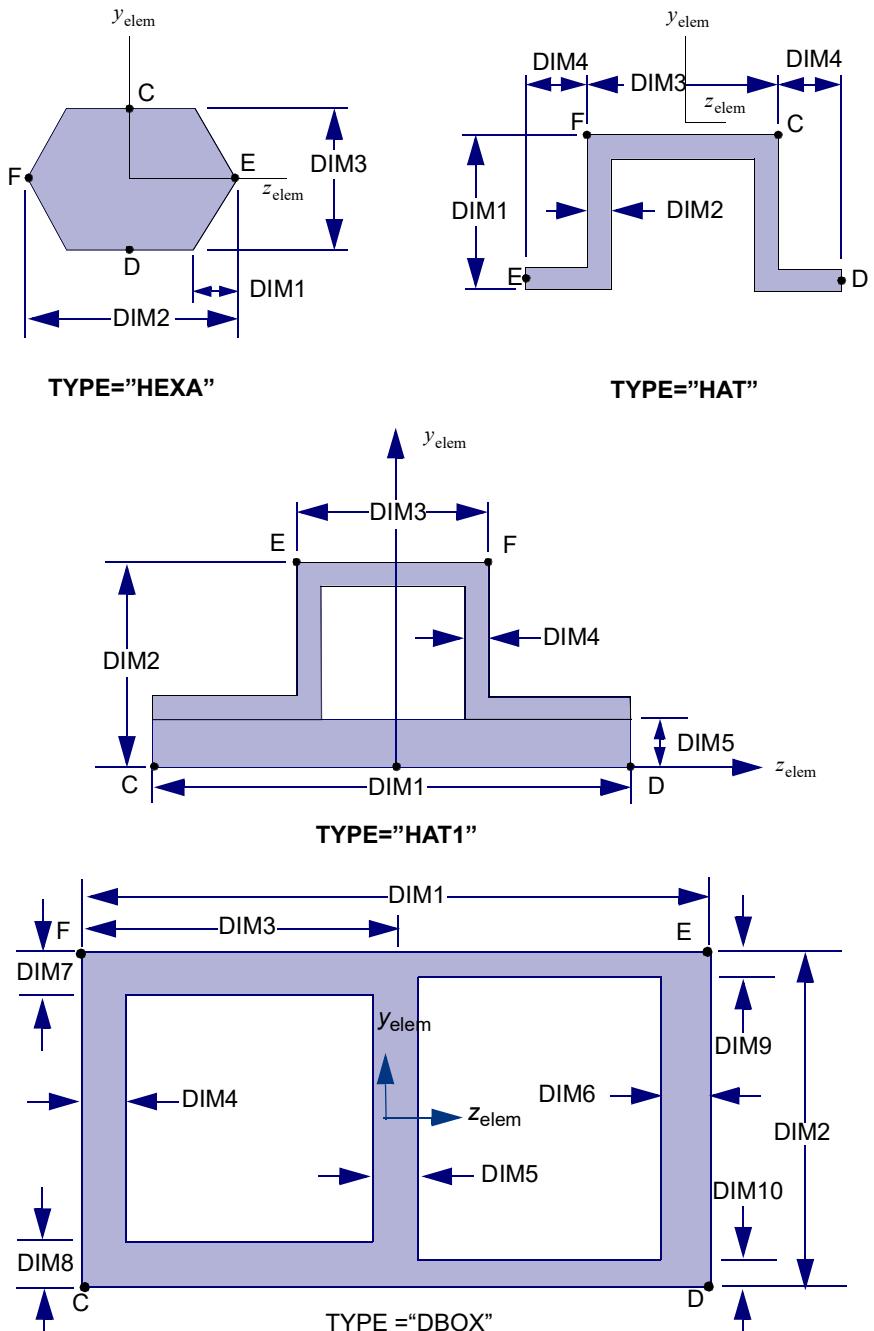


Figure 9-122 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

**PBARN1**

## Nonlinear Property Extensions for a PBAR or PBARL Entry

Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry in SOL 400.

**Format:**

| 1      | 2    | 3    | 4    | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|------|------|------|------|---|---|---|---|----|
| PBARN1 | PID  | MID  |      | SECT |   |   |   |   |    |
|        | “C2” | BEH2 | INT2 |      |   |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PBARN1 | 29 | 73 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number of an existing PBAR entry. (Integer > 0,)   |
| MID        | Material ID. Remark 10. (Integer $\geq 0$ )  |
| SECT       | Section integration. SECT = “S” a smeared cross section is used for integration. SECT = “N” a numerically integrated cross section is used. See Remark 7. (Character Default S or blank) |
| C2         | Keyword indicating that items following apply to elements with two end grids. (Character)  |
| BEH2       | Element structural behavior. See Remark 4. (Character Default BAR)   |
| INT2       | Integration scheme. See Remarks 4. and 5. (Character Default LC)   |

**Remarks:**

1. The PID above must point to an existing PBAR or PBARL Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PBAR or PBARL entry. If  $> 0$  it will override the MID value on the PBAR entry.
3. The MID entry may point to the MAT1 entry. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

| Implicit Structural Materials |
|-------------------------------|
| MAT1                          |
| MATVE                         |
| MATVP                         |
| MATEP                         |

| Implicit Structural Materials |
|-------------------------------|
| MATF                          |
| MATS1                         |
| MATSMA                        |

4. BEH2 refers to the nonlinear structural behavior of the BAR element. An underlined item delineates a default.

| Structural Classification of Elements |            |                        |
|---------------------------------------|------------|------------------------|
| Element Structural Type               | BEHAV CODE | Integration Code       |
| Bar                                   | BAR        | <u>LC</u><br>LCC<br>LS |

5. Integration codes in Remark 4. are:

| INT CODE | Integration Type            |
|----------|-----------------------------|
| LC       | Linear/Cubic                |
| LCC      | Linear/Cubic Closed section |
| LS       | Linear-shear                |

6. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.

## 7. Smeared section support.

| Primary BAR Property | INT CODE | SECT | Interpolation  | Usage  | COMMENTS   |
|----------------------|----------|------|--|--|--|
| PBAR                 | LC       | S    | Linear<br>Interpolation for axial displacement and Cubic<br>Interpolation for transverse displacements | Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam.  | Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.  |
|                      | LS       | S    | Linear<br>Interpolation for axial, transverse displacements and rotations                              | Both Thin-Walled and Solid Sections with elastic materials.<br>Captures transverse shear effects - useful for deep beams | Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.  |
|                      | LC or LS | N    |  |  | Not Supported  |
|                      | LCC      | S    |  |  | Not Supported  |
|                      | LCC      | N    |  |  | Not Supported  |
| PBART                | LC or LS | S    |  |  | Not Supported  |
|                      | LC       | N    | Linear<br>Interpolation for axial displacement and Cubic<br>Interpolation for transverse displacements | Solid Sections. Euler-Bernoulli Beam   | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials are supported. MATHE/MATHP are not supported. |

| Primary BAR Property | INT CODE | SECT | Interpolation  | Usage  | COMMENTS   |
|----------------------|----------|------|--|--|--|
|                      | LS       | N    | Linear<br>Interpolation for axial displacements, transverse displacements and rotations                | Solid Sections.<br>Captures transverse shear effects - useful for deep beams | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported. |
|                      | LCC      | S    |  |  | Not Supported  |
|                      | LCC      | N    | Linear<br>Interpolation for axial displacement and Cubic<br>Interpolation for transverse displacements | Thin-Walled Sections.<br>Euler-Bernoulli Beam.                               | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported. |

8. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
9. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
10. The structural element damping coefficient, GE, is not supported on elements which reference PBARN1.

**PBCOMP**

## Beam Property (Alternate Form of PBEAM)

Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. This entry is also used to specify lumped areas of the beam cross section for nonlinear analysis and/or composite analysis.

**Format:**

| 1      | 2      | 3   | 4  | 5    | 6  | 7   | 8      | 9   | 10 |
|--------|--------|-----|----|------|----|-----|--------|-----|----|
| PBCOMP | PID    | MID | A  | I1   | I2 | I12 | J      | NSM |    |
|        | K1     | K2  | M1 | M2   | N1 | N2  | SYMOPT |     |    |
|        | Y1     | Z1  | C1 | MID1 |    |     |        |     |    |
|        | Y2     | Z2  | C2 | MID2 |    |     |        |     |    |
|        | -etc.- |     |    |      |    |     |        |     |    |

**Example:**

|        |      |     |      |    |  |  |   |  |  |
|--------|------|-----|------|----|--|--|---|--|--|
| PBCOMP | 39   | 6   | 2.9  |    |  |  |   |  |  |
|        |      |     |      |    |  |  | 1 |  |  |
|        | -0.5 | 1.2 | 0.1  | 18 |  |  |   |  |  |
|        | 0.2  | 0.9 | 0.15 |    |  |  |   |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. See Remark 1. (Integer > 0)  |
| MID        | Material identification number. See Remarks 3. and 6. (Integer > 0)  |
| A          | Area of beam cross section. (Real > 0.0)   |
| I1         | Area moment of inertia in plane 1 about the neutral axis. See Remark 7. (Real > 0.0)   |
| I2         | Area moment of inertia in plane 2 about the neutral axis. See Remark 7. (Real > 0.0)   |
| I12        | Area product of inertia. See Remark 7. (Real; Default = 0.0, but $I1 \cdot I2 - (I12)^2 > 0.0$ )   |
| J          | Torsional stiffness parameter. See Remark 7. (Real > 0.0; Default = 0.0)   |
| NSM        | Nonstructural mass per unit length. (Real > 0.0; Default = 0.0)  |
| K1, K2     | Shear stiffness factor K in $K^*A^*G$ for plane 1 and plane 2. See Remark 5. (Real > 0.0; Default = 1.0)   |
| M1, M2     | The (y,z) coordinates of center of gravity of nonstructural mass. See the figure in the CBEAM entry description. (Real; Default = 0.0)           |
| N1, N2     | The (y,z) coordinates of neutral axis. See the figure in the CBEAM entry description. (Real; Default = 0.0)                                      |
| SYMOPT     | Symmetry option to input lumped areas for the beam cross section. See Figure 9-124 and Remark 8. ( $0 \leq \text{Integer} \leq 5$ ; Default = 0) |

| Descriptor | Meaning  |
|------------|--|
| Yi, Zi     | The (y,z) coordinates of the lumped areas in the element coordinate system. See Remark 1. (Real) |
| Ci         | Fraction of the total area for the i-th lumped area. (Real > 0.0; Default = 0.0)                 |
| MIDi       | Material identification number for the i-th integration point. See Remark 6. (Integer > 0)       |

**Remarks:**

1. PBCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
2. The second continuation entry may be repeated 18 more times. If SYMOPT = 5 a maximum of 21 continuation entries is allowed; i.e., a maximum of 20 lumped areas may be input. If SYMOPT = 1 through 4, the total number of areas input plus the total number generated by symmetry must not exceed 20. If these are not specified, the program defaults, as usual, to the elliptically distributed eight nonlinear rods. See [Figure 9-123](#).

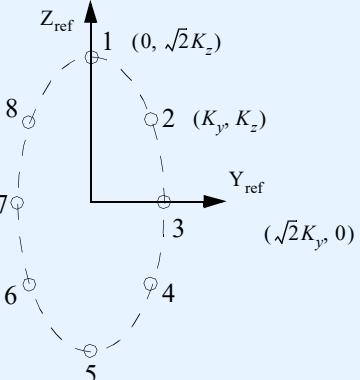
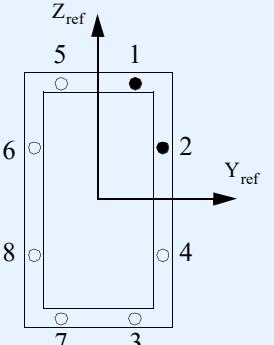
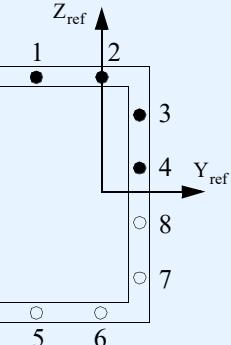
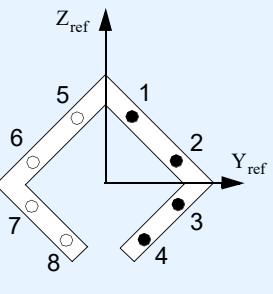
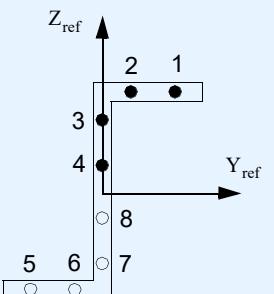
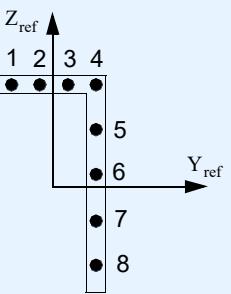
|  |  |   |
|--|--|---|
|  <p><u>SYMOPT = 0</u><br/>No continuation entries<br/>Symmetric about <math>Y_{ref}</math> and <math>Z_{ref}</math></p> $K_y = \sqrt{\frac{I_{zz}}{A}}, K_z = \sqrt{\frac{I_{yy}}{A}}, C1 = \frac{1}{8}$ <p><math>I_{zz}</math> - moment of inertia about z-axis<br/><math>I_{yy}</math> - moment of inertia about y-axis</p> |  <p><u>SYMOPT = 1</u><br/>(w/continuation entries)<br/>Symmetric about <math>Y_{ref}</math> and <math>Z_{ref}</math></p> <p><math>Y1 = Y3 = -Y5 = -Y7</math><br/><math>Z1 = -Z3 = Z5 = -Z7</math>, etc.</p> |  <p><u>SYMOPT = 2</u><br/>Symmetric about <math>Y_{ref}</math></p> <p><math>Y1 = Y5</math><br/><math>Z1 = -Z5</math>, etc.</p> |
|  <p><u>SYMOPT = 3</u><br/>Symmetric about <math>Z_{ref}</math></p> <p><math>Y1 = -Y5, Z1 = Z5</math>, etc.</p>   |  <p><u>SYMOPT = 4</u><br/>Mirror Symmetry about <math>Y_{ref}</math> and <math>Z_{ref}</math></p> <p><math>Y1 = -Y5, Z1 = -Z5</math>, etc.</p>   |  <p><u>SYMOPT = 0 or 5</u><br/>No symmetry</p>  |

Figure 9-123 PBCOMP Entry SYMOPT Type Examples with 8 Lumped Areas

**Notes:**

Integration points (lumped area) are numbered 1 through 8.

User-specified points are denoted by ● and the mirrored points are denoted by ○.

3. For structural problems, MID and MIDi must reference a MAT1 material entry. For material nonlinear analysis, the material should be perfectly plastic since the plastic hinge formulation is not valid for strain hardening. For heat transfer problems, MID and MIDi must reference a MAT4 or MAT5 material entry.
4. For the case where the user specifies I1, I2 and I12 on the parent entry, the stress-output location may also be specified on continuation entries. The (y,z) coordinates specified on these entries will serve as stress output locations with the corresponding Ci's set to 0. Stress output is provided at the first four lumped area locations only. If one of the symmetry options is used and fewer than four lumped areas are input explicitly, the sequence of output locations in the imaged quadrants is shown in [Figure 9-123](#). For one specific example in the model shown in Remark 8. ([Figure 9-124](#)), output can be obtained at points 1 and 2 and in the image points 3 and 4.
5. Blank fields for K1 and K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear stiffness becomes rigid and the transverse shear flexibilities are set to 0.0.
6. The values  $E_0$  and  $G_0$  are computed based on the value of MID on the parent entry. MID is will follow the same symmetry rules as Ci depending on the value of SYMOPT. If the MIDi field on a continuation entry is blank, the value will be that of MID on the parent entry. MIDi values may be input on continuations without the corresponding Yi, Zi, and Ci values to allow different stress-strain laws.
7. If the lumped cross-sectional areas are specified, fields I1, I2, and I12 will be ignored. These and other modified values will be calculated based on the input data (Yi, Zi, Ci, MIDi) as follows:

$$y_{NA} = \frac{\sum_{i=1}^n Y_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$z_{NA} = \frac{\sum_{i=1}^n Z_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$\bar{A} = A \sum_{i=1}^n \frac{C_i E_i}{E_0}$$

$$\bar{I}_1 = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})^2}{E_o}$$

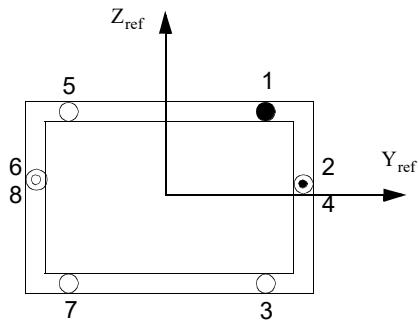
$$\bar{I}_2 = A \sum_{i=1}^n \frac{C_i E_i (Z_i - z_{NA})^2}{E_o}$$

$$\bar{I}_{12} = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})(Z_i - z_{NA})}{E_o}$$

$$J = J \sum_{i=1}^n \frac{C_i G_i}{G_o}$$

where n is the number of lumped cross-sectional areas specified.

8. For a doubly symmetric section (SYMOPT = 1), if the lumped areas are specified on either axis, the symmetry option will double the areas. For example, for the section shown in [Figure 9-124](#), points 2 and 4 are coincident and so are points 6 and 8. In such cases, it is recommended that users input the value of area as half of the actual value at point 2 to obtain the desired effect.



**Figure 9-124 Doubly Symmetric PBCOMP Section**

9. For SYMOPT =0, 5, or blank at least three Yi and one Zi coordinates must be nonzero. All of the points (input plus mirrored) should not lie on a straight line. All of the points (input plus mirrored) should not lie on a straight line or a FATAL message will be issued.

**PBEAM****Beam Property**

Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

**Format:**

| 1     | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|-------|--------|--------|--------|--------|--------|--------|--------|--------|----|
| PBEAM | PID    | MID    | A(A)   | I1(A)  | I2(A)  | I12(A) | J(A)   | NSM(A) |    |
|       | C1 (A) | C2 (A) | D1 (A) | D2 (A) | E1 (A) | E2 (A) | F1 (A) | F2 (A) |    |

The next two continuations are repeated for each intermediate station as described in Remark 6. and SO and X/XB must be specified.

|  |    |      |    |    |    |     |    |     |  |
|--|----|------|----|----|----|-----|----|-----|--|
|  | SO | X/XB | A  | I1 | I2 | I12 | J  | NSM |  |
|  | C1 | C2   | D1 | D2 | E1 | E2  | F1 | F2  |  |

The last two continuations are:

|  |       |       |       |       |        |        |       |       |  |
|--|-------|-------|-------|-------|--------|--------|-------|-------|--|
|  | K1    | K2    | S1    | S2    | NSI(A) | NSI(B) | CW(A) | CW(B) |  |
|  | M1(A) | M2(A) | M1(B) | M2(B) | N1(A)  | N2(A)  | N1(B) | N2(B) |  |

**Example:**

Tapered beam with A=2.9 at end A and A=5.3 at end B.

|       |     |     |     |      |      |  |      |  |  |
|-------|-----|-----|-----|------|------|--|------|--|--|
| PBEAM | 39  | 6   | 2.9 | 3.5  | 5.97 |  |      |  |  |
|       |     |     | 2.0 | -4.0 |      |  |      |  |  |
|       | YES | 1.0 | 5.3 | 56.2 | 78.6 |  |      |  |  |
|       |     |     | 2.5 | -5.0 |      |  |      |  |  |
|       |     |     | 1.1 |      | 2.1  |  | 0.21 |  |  |
|       |     |     |     |      | 0.5  |  | 0.0  |  |  |

| Descriptor | Meaning  | Default Values |
|------------|--|----------------|
| PID        | Property identification number. (Integer > 0)  | Required       |
| MID        | Material identification number. See Remarks 1. and 3.<br>(Integer > 0)                                     | Required       |
| A(A)       | Area of the beam cross section at end A. (Real > 0.0)  | Required       |
| I1(A)      | Area moment of inertia at end A for bending in plane 1 about the neutral axis. See Remark 10. (Real > 0.0) | Required       |
| I2(A)      | Area moment of inertia at end A for bending in plane 2 about the neutral axis. See Remark 10. (Real > 0.0) | Required       |

| Descriptor      | Meaning   | Default Values   |
|-----------------|---|--|
| I12(A)          | Area product of inertia at end A. See Remark 10.<br>(Real, but $I_1 \cdot I_2 - (I_{12})^2 > 0.0$ )   | 0.0  |
| J(A)            | Torsional stiffness parameter at end A. See Remark 10.<br>(Real $\geq 0.0$ but $> 0.0$ if warping is present)   | Default= $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences |
| NSM(A)          | Nonstructural mass per unit length at end A. (Real)   | 0.0  |
| Ci(A), Di(A)    | The y and z locations ( $i = 1$ corresponds to y and $i = 2$ corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery. (Real)  | $y = z = 0.0$  |
| Ei(A), Fi(A)    |   |  |
| SO              | Stress output request option. See Remark 9.<br>(Character)  | Required*  |
|                 | “YES” Stresses recovered at points Ci, Di, Ei, and Fi on the next continuation.   |  |
|                 | “YESA” Stresses recovered at points with the same y and z location as end A.  |  |
|                 | “NO” No stresses or forces are recovered.   |  |
| X/XB            | Distance from end A in the element coordinate system divided by the length of the element See Figure 9-125 in Remark 10. (Real, $0.0 < x/xb \leq 1.0$ )   | Required*<br>See Remark 6.   |
| A, I1, I2, I12, | Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x. (Real; J > 0.0 if warping is present.)  | See Remark 7.  |
| J, NSM          |   |  |
| Ci, Di, Ei, Fi  | The y and z locations ( $i = 1$ corresponds to y and $i = 2$ corresponds to z) in element coordinates relative to the shear center (see Figure 9-125 in Remark 10.) for the cross section located at X/XB. The values are fiber locations for stress data recovery. Ignored for beam p-elements. (Real) |  |
| K1, K2          | Shear stiffness factor K in $K^*A^*G$ for plane 1 and plane 2. See Remark 12. (Real)  | 1.0, 1.0   |
| S1, S2          | Shear relief coefficient due to taper for plane 1 and plane 2. Ignored for beam p-elements. (Real)  | 0.0, 0.0   |
| NSI(A), NSI(B)  | Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B. See Figure 9-125. (Real)  | 0.0, same as end A   |

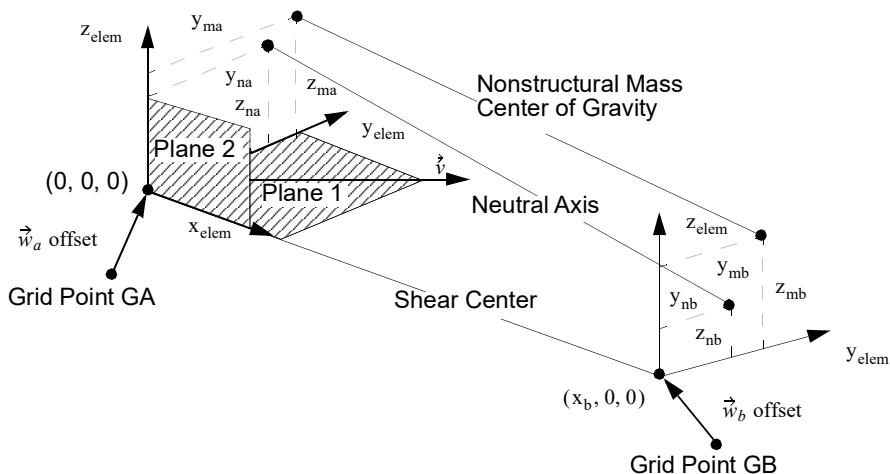
| Descriptor                    | Meaning   | Default Values  |
|-------------------------------|---|---|
| CW(A), CW(B)                  | Warping coefficient for end A and end B. Ignored for beam p-elements. See Remark 11. (Real)                                 | 0.0, same as end A                                      |
| M1(A), M2(A),<br>M1(B), M2(B) | (y,z) coordinates of center of gravity of nonstructural mass for end A and end B. See <a href="#">Figure 9-125</a> . (Real) | 0.0 (no offset from shear center), same values as end A |
| N1(A), N2(A),<br>N1(B), N2(B) | (y,z) coordinates of neutral axis for end A and end B. See <a href="#">Figure 9-125</a> . (Real)                            | 0.0 (no offset from shear center), same values as end A |

**Remarks:**

- For structural analysis, MID must reference a MAT1 material entry (SOL 600). The beam may be described by any valid stress-strain law. A plastic hinge is not used for SOL 600 and SOL 700; instead, a standard nonlinear analysis is performed.
- For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic; for example, H = 0.0 on the MATS1 entry. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.
- For heat transfer analysis, MID must reference a MAT4 or MAT5 material entry.
- If no stress data at end A is to be recovered and a continuation with the SO field is specified, then the first continuation entry, which contains the fields C1(A) through F2(A), may be omitted.
- If SO is “YESA” or “NO”, the third continuation entry, which contains the fields C1 through F2, must be omitted. If SO is “YES”, the continuation for Ci, Di, Ei, and Fi must be the next entry. The blank fields are defaulted to 0.0 on these continuations.
- The rules for the continuations entries are:
  - The second and third continuation entries, which contain fields SO through F2, may be repeated nine more times for intermediate X/XB values for linear beam elements. The order of these continuation pairs is independent of the X/XB value; however, one value of X/XB must be 1.0, corresponding to end B. The intermediate stress output requests will be ignored in the nonlinear solution sequences (SOLs 106 and 129).
  - The value of X/XB must be unique among the stations of a PBEAM. Duplication of X/XB is not permitted.
  - The fourth and fifth continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.
- If any fields 4 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and NSM are set to the values given for end A. For the continuations that have intermediate values of X/XB between 0.0 and 1.0 and use the default option (any of the fields 4 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing section properties.

8. Blank fields for K1, K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear flexibilities are set to 0.0 and field G on the MAT1 entry selected by MID must be nonzero.
9. If end B forces are desired and station data are input, use “YES” or “YESA” in the SO field of the record with X/XB=1. If station data are not input you will get end B forces if forces are requested.
10. [Figure 9-125](#) describes the PBEAM element coordinate system.

$$\begin{array}{lll}
 I_{11} = I_{(zz)_{na}} & N1(A) = y_{na} & N1(B) = y_{nb} \\
 I_{12} = I_{(yy)_{na}} & N2(A) = z_{na} & N2(B) = z_{nb} \\
 I_{12} = I_{(zy)_{na}} & M1(A) = y_{ma} & M1(B) = y_{mb} \\
 J = I_{(xx)_{na}} & M2(A) = z_{ma} & M2(B) = z_{mb}
 \end{array}$$



[Figure 9-125](#) PBEAM Element Coordinate System

11. The warping coefficient CW is represented in the following differential equation for the torsion of a beam about the axis of the shear centers:

$$G \frac{d}{dx} \left( J \frac{d\theta}{dx} \right) - E \frac{d^2}{dx^2} \left( CW \frac{d^2\theta}{dx^2} \right) = m$$

where:

G = shear modulus  
 J = torsional stiffness  
 E = Young's modulus  
 $\theta$  = angle of rotation at any cross-section  
 m = applied torsional moment per unit length

**Note:** CW has units of (length)<sup>6</sup>.

12. The shear stiffness factors  $K_1$  and  $K_2$  adjust the effective transverse shear cross-section area according to the Timoshenko beam theory. Their default values of 1.0 approximate the effects of shear deformation. To neglect shear deformation (i.e., to obtain the Bernoulli-Euler beam theory), the values of  $K_1$  and  $K_2$  should be set to 0.0.
13. In nonlinear analysis the location of the 8 plastic rods is the same on the PBEAM entry as it is on the SYMOPT field on the PBCOMP entry when SYMOPT = 0. Please see the [Figure 9-123](#) for more information on the SYMOPT field.
14. For SOL 600, for structural analysis, MID must reference a MAT1 material entry.
15. RC network solver only supports constant cross section beam for thermal analysis.
16. PBEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

**PBEAM3**

## Three-node Beam Property

Defines the properties of a three-node beam element (CBEAM3 entry).

**Format:**

| 1      | 2        | 3       | 4       | 5        | 6       | 7       | 8        | 9       | 10 |
|--------|----------|---------|---------|----------|---------|---------|----------|---------|----|
| PBEAM3 | PID      | MID     | A(A)    | IZ(A)    | IY(A)   | IYZ(A)  | J(A)     | NSM(A)  |    |
|        | CY(A)    | CZ(A)   | DY(A)   | DZ(A)    | EY(A)   | EZ(A)   | FY(A)    | FZ(A)   |    |
|        | SO(B)    |         | A(B)    | IZ(B)    | IY(B)   | IYZ(B)  | J(B)     | NSM(B)  |    |
|        | CY(B)    | CZ(B)   | DY(B)   | DZ(B)    | EY(B)   | EZ(B)   | FY(B)    | FZ(B)   |    |
|        | SO(C)    |         | A(C)    | IZ(C)    | IY(C)   | IYZ(C)  | J(C)     | NSM(C)  |    |
|        | CY(C)    | CZ(C)   | DY(C)   | DZ(C)    | EY(C)   | EZ(C)   | FY(C)    | FZ(C)   |    |
|        | KY       | KZ      | NY(A)   | NZ(A)    | NY(B)   | NZ(B)   | NY(C)    | NZ(C)   |    |
|        | MY(A)    | MZ(A)   | MY(B)   | MZ(B)    | MY(C)   | MZ(C)   | NSIY(A)  | NSIZ(A) |    |
|        | NSIYZ(A) | NSIY(B) | NSIZ(B) | NSIYZ(B) | NSIY(C) | NSIZ(C) | NSIYZ(C) | CW(A)   |    |
|        | CW(B)    | CW(C)   | STRESS  |          |         |         |          |         |    |
|        | WC(A)    | WYC(A)  | WZC(A)  | WD(A)    | WYD(A)  | WZD(A)  | WE(A)    | WYE(A)  |    |
|        | WZE(A)   | WF(A))  | WYF(A)  | WZF(A)   | WC(B)   | WYC(B)  | WZC(B)   | WD(B)   |    |
|        | WYD(B)   | WZD(B)  | WE(B)   | WYE(B)   | WZE(B)  | WF(B)   | WYF(B)   | WZF(B)  |    |
|        | WC(C)    | WYC(C)  | WZC(C)  | WD(C)    | WYD(C)  | WZD(C)  | WE(C)    | WYE(C)  |    |
|        | WZE(C)   | WF(C)   | WYF(C)  | WZF(C)   |         |         |          |         |    |

**Example:**

|        |      |     |      |      |      |     |  |     |  |
|--------|------|-----|------|------|------|-----|--|-----|--|
| PBEAM3 | 1010 | 2   | 2.9  | 3.5  | 5.97 |     |  | 1.0 |  |
|        | 0.2  | 3.0 | -1.2 | 2.6  | 2.0  | 0.5 |  |     |  |
|        | YES  |     | 1.0  | 23.6 | 34.7 |     |  |     |  |
|        | 1.1  | 3.2 |      |      |      |     |  |     |  |
|        | YESA |     | 3.2  | 2.1  | 3.2  |     |  | 1.0 |  |
|        | 0.8  |     | 0.5  |      |      |     |  |     |  |
|        | 0.9  | 1.0 |      | 1.5  |      |     |  |     |  |
|        |      | 1.0 |      |      |      |     |  |     |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0; Required)   |
| MID        | Material identification number. See Remark 1. (Integer > 0; Required)                           |
| A(A)       | Area of the beam cross-section at end A. (Real > 0.0; Required)                                 |
| IZ(A)      | Area moment of inertia at end A about local z-axis and the neutral axis. (Real > 0.0; Required) |

| Descriptor                                    | Meaning  |
|---|--|
| IY(A)   | Area moment of inertia at end A about local y-axis and the neutral axis. (Real > 0.0; Required)  |
| IYZ(A)  | Area product of inertia at end A about local y- and z-axes and the neutral axis. If y- and z- axes are principal axes, then IYZ(A)=0.0. (Real, but $I_y \cdot I_z - I_{yz}^2 > 0.0$ ; Default = 0.0)   |
| J(A)  | Torsional stiffness parameter at end A. (Real > 0.0; Default = IZ+IY)  |
| NSM(A)  | Nonstructural mass per unit length at end A. (Real; Default = 0.0)   |
| $Ci(j)$ , $Di(j)$<br>$Ei(j)$ , $Fi(j)$        | The local y and z coordinates ( $i=Y, Z$ ) at point $j$ ( $j=A, B, C$ ), used for stress output. (Real; Default = 0.0)   |
| A(j), IZ(j), IY(j)<br>IYZ(j), J(j),<br>NSM(j) | Area, moments of inertia, torsional stiffness parameter and nonstructural mass for the cross-section at $j$ ( $j=B, C$ ). (Real; See Remark 2.)  |
| SO(j)   | Stress output request option at $j$ ( $j=B, C$ ). (Character; Default = "YESA") <ul style="list-style-type: none"> <li>"YES"      Stresses are recovered at <math>Ci</math>, <math>Di</math>, <math>Ei</math>, and <math>Fi</math> on the next continuation.</li> <li>"YESA"     Stresses are recovered at points with the same (<math>y, z</math>) location at end A</li> </ul> |
| KY, KZ  | Shear effectiveness factors for local y- and z-directions. (Real > 0.0, Default = 1.0)   |
| NY(j), NZ(j)                                  | Local ( $y, z$ ) coordinates of neutral axis for $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$ )  |
| MY(j), MZ(j)                                  | Local ( $y, z$ ) coordinates of nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B,C$ )  |
| NSIY(j), NSIZ(j)                              | Nonstructural mass moments of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$ )  |
| NSIYZ(j)                                      | Nonstructural mass product of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$ )  |
| CW(j)   | Warping coefficient at $j$ ( $j=A, B, C$ ). (Real $\geq 0.0$ ; Default = 0.0 at end A; same values as end A for $j = B, C$ )   |
| STRESS  | Location selection for stress, strain and force output. (Character; Default = "GRID"; See Remark 3.)   |
| Wi(j)   | Values of warping function at stress recovery points $i = C, D, E$ and $F$ , at location $j=A, B$ , and $C$ . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$ )  |
| WYi(j), WZi(j)                                | Gradients of warping function in the local ( $y, z$ ) coordinate system at stress recovery points $i=C, D, E$ , and $F$ , at location $j=A, B$ , and $C$ . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$ .)  |

**Remarks:**

1. For structural analysis, MID must reference a MAT1, MAT2 or MAT8 material entry.
2. If any fields 4 through 9, for values of A, IZ, IY, IYZ, J and NSM at end B or C, are blank, then those values for end B or C are set to the values given for end A.
3. If STRESS=“GRID”, then the stresses, strains and forces are recovered at A, B and C. If STRESS=“GAUSS”, then the stresses, strains and forces are recovered at Gauss integration points,  $\xi = \{1/\sqrt{3}, 1/\sqrt{3}, 0\}$ . The beam cross-section properties at these points are interpolated from those at A, B and C.
4. If all fields of  $W_i(j)$ ,  $WY_i(j)$  and  $WZ_i(j)$  ( $i=C, D, E, F$  and  $j=A, B, C$ ), are left blank, both stresses and strains due to the warping effect will not be recovered at the stress recovery points.
5. PBEAM3 is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM3 property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
6. When PBMSECT is used the resulting PBEAM3 image printed after the:

\*\*\* USER INFORMATION MESSAGE 4379 (IFP9B)

THE USER SUPPLIED PBMSECT BULK DATA ENTRIES ARE REPLACED BY THE FOLLOWING PBEAM3 ENTRIES.

contains special composite related data and will look different from the above entry description.

## PBEAMD

PBEAMD usage is no longer recommended and will be removed in a future version. Use PBEAM, PBDISCR, PBSPOT, PBEAM71 instead.

**PBEAML****Beam Cross-Section Property**

Defines the properties of a beam element by cross-sectional dimensions.

**Format:**

(Note: n = number of dimensions and m = number of intermediate stations)

| 1      | 2       | 3       | 4       | 5       | 6       | 7       | 8       | 9       | 10 |
|--------|---------|---------|---------|---------|---------|---------|---------|---------|----|
| PBEAML | PID     | MID     | GROUP   | TYPE    |         |         |         |         |    |
|        | DIM1(A) | DIM2(A) | -etc.-  | DIMn(A) | NSM(A)  | SO(1)   | X(1)/XB | DIM1(1) |    |
|        | DIM2(1) | -etc.-  | DIMn(1) | NSM(1)  | SO(2)   | X(2)/XB | DIM1(2) | DIM2(2) |    |
|        | -etc.-  | DIMn(2) | NSM(m)  | -etc.-  | SO(m)   | X(m)/XB | DIM1(m) | -etc.-  |    |
|        | DIMn(m) | NSM(m)  | SO(B)   | 1.0     | DIM1(B) | DIM2(B) | -etc.-  | DIMn(B) |    |
|        | NSM(B)  |         |         |         |         |         |         |         |    |

**Example:**

|        |     |      |     |     |     |     |     |     |  |
|--------|-----|------|-----|-----|-----|-----|-----|-----|--|
| PBEAML | 99  | 21   |     | T   |     |     |     |     |  |
|        | 12. | 14.8 | 2.5 | 2.6 |     | NO  | 0.4 | 6.  |  |
|        | 7.  | 1.2  | 2.6 |     | YES | 0.6 | 6.  | 7.8 |  |
|        | 5.6 | 2.3  |     | YES |     |     |     |     |  |

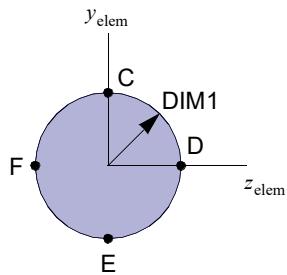
| Descriptor  | Meaning   |
|-------------|---|
| PID         | Property identification number. (Integer > 0)   |
| MID         | Material identification number. (Integer > 0)   |
| GROUP       | Cross-section group. (Character; Default = "MSCBML0")   |
| TYPE        | Cross-section shape. See Remark 4. (Character: "ROD", "TUBE", "TUBE2", "L", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP = "MSCBML0") |
| DIMi(j)     | Cross-section dimensions at end A, intermediate station j and end B. (Real > 0.0 for GROUP = "MSCBML0")   |
| NSM(j)      | Nonstructural mass per unit length. (Default = 0.0)   |
| SO(j),SO(B) | Stress output request option for intermediate station j and end B. (Character; Default = "YES")   |
| YES         | Stresses recovered at all points on next continuation and shown in as C, D, E, and F.   |

| Descriptor | Meaning  |
|------------|--|
| NO         | No stresses or forces are recovered.   |
| X(j)/XB    | Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (REAL, 0.0<x(j)/xb ≤1.0, default=1.0) |

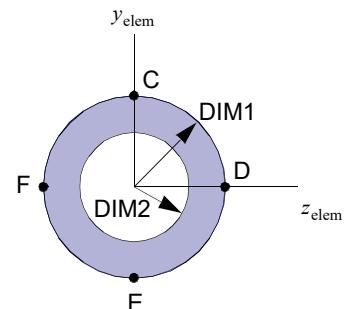
**Remarks:**

1. For structural problems, PBEAML entries must reference a MAT1 material entry.
2. PBEAML is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAML property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
4. See the PBEAM entry description for a discussion of beam-element geometry.
5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j)/XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.
6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion of how to include a user-defined beam library.
7. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in . The element coordinate system is located at the shear center.
8. A function of this entry is to derive an equivalent PBEAM entry. Any sorted echo request will also cause printout and/or punch of the derived PBEAM.
9. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words.  
None of these limits are exceeded with the MSC beam library, but a user defined beam library could.
10. The finite element formulation (FEF) utilized for the arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBEAML when GROUP=MSCBML0. The original beam equations, which are based on thin-walled assumption can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’.
11. For optimization, individual DIMx of PBEAML can be selected as designed properties even with finite element formulation.

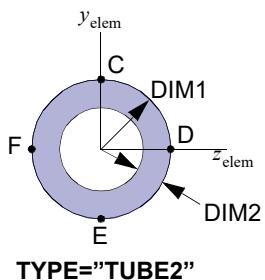
12. The origin of the  $y_{elem}$  and  $z_{elem}$  coordinate system for the section types is at the shear center for all element types when using the FEF. For section types 'L', 'T', 'T1', 'T2' and 'HAT1', no shear center calculation is calculated with the original beam equations and the origins of these types are at the locations specified in the following figures. For these sections types, this can result in a difference in the stress recovery points and the neutral axis locations between the beam equations and the FEF. The FEF results are considered more exact. For the remaining types with the beam equation, the shear center is either computed or is obvious due to symmetry considerations.



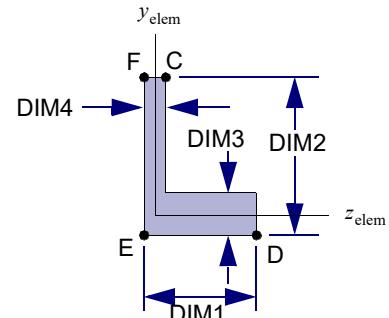
**TYPE="ROD"**



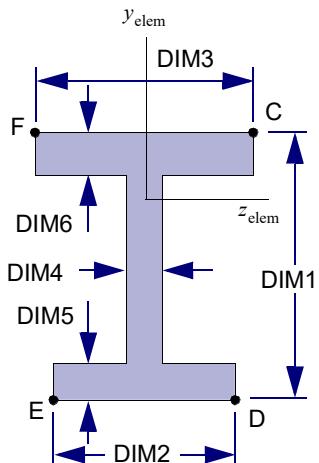
**TYPE="TUBE"**



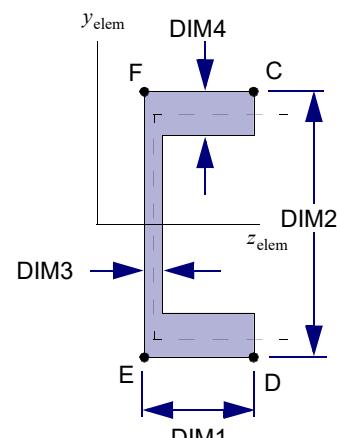
**TYPE="TUBE2"**



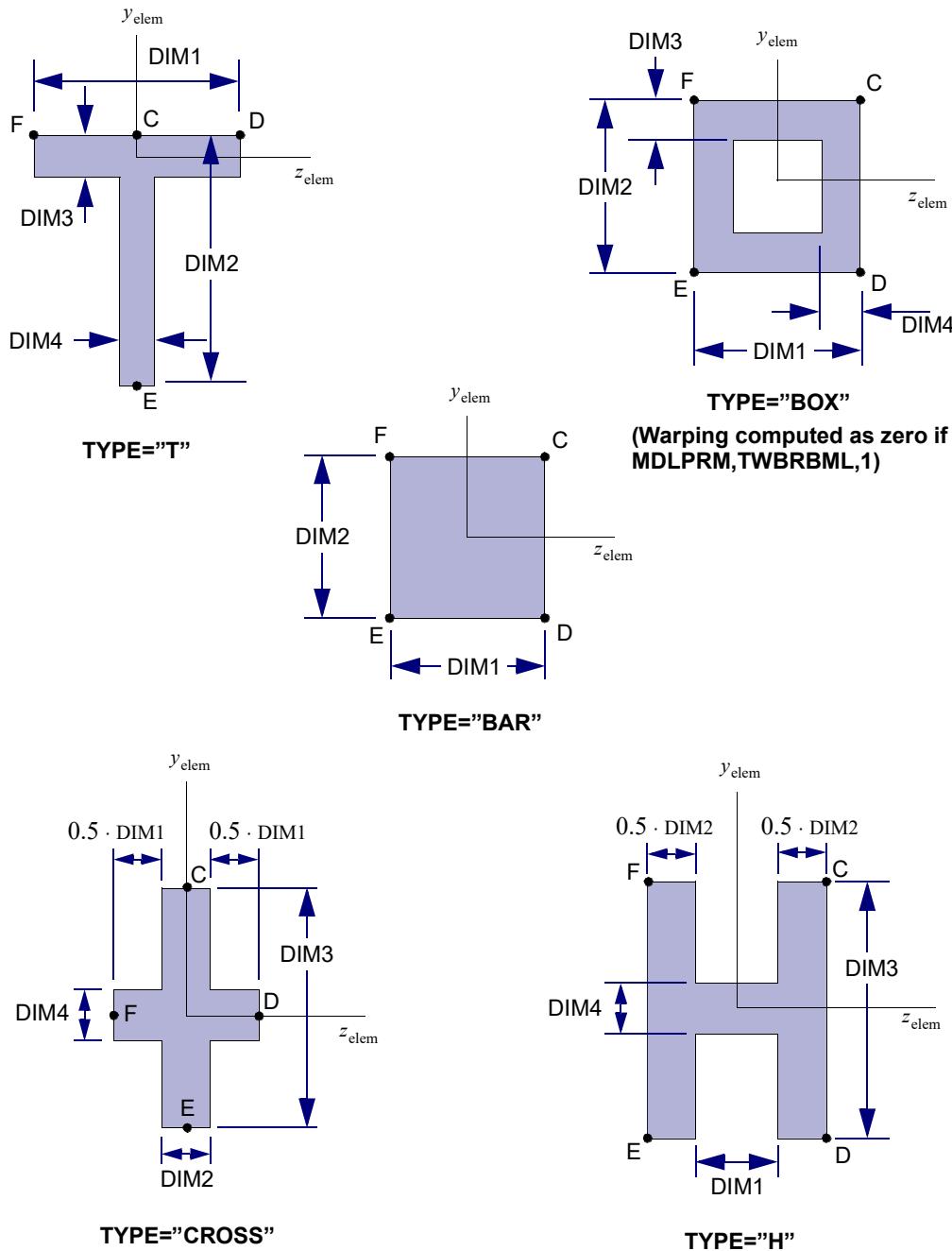
**TYPE="L"**

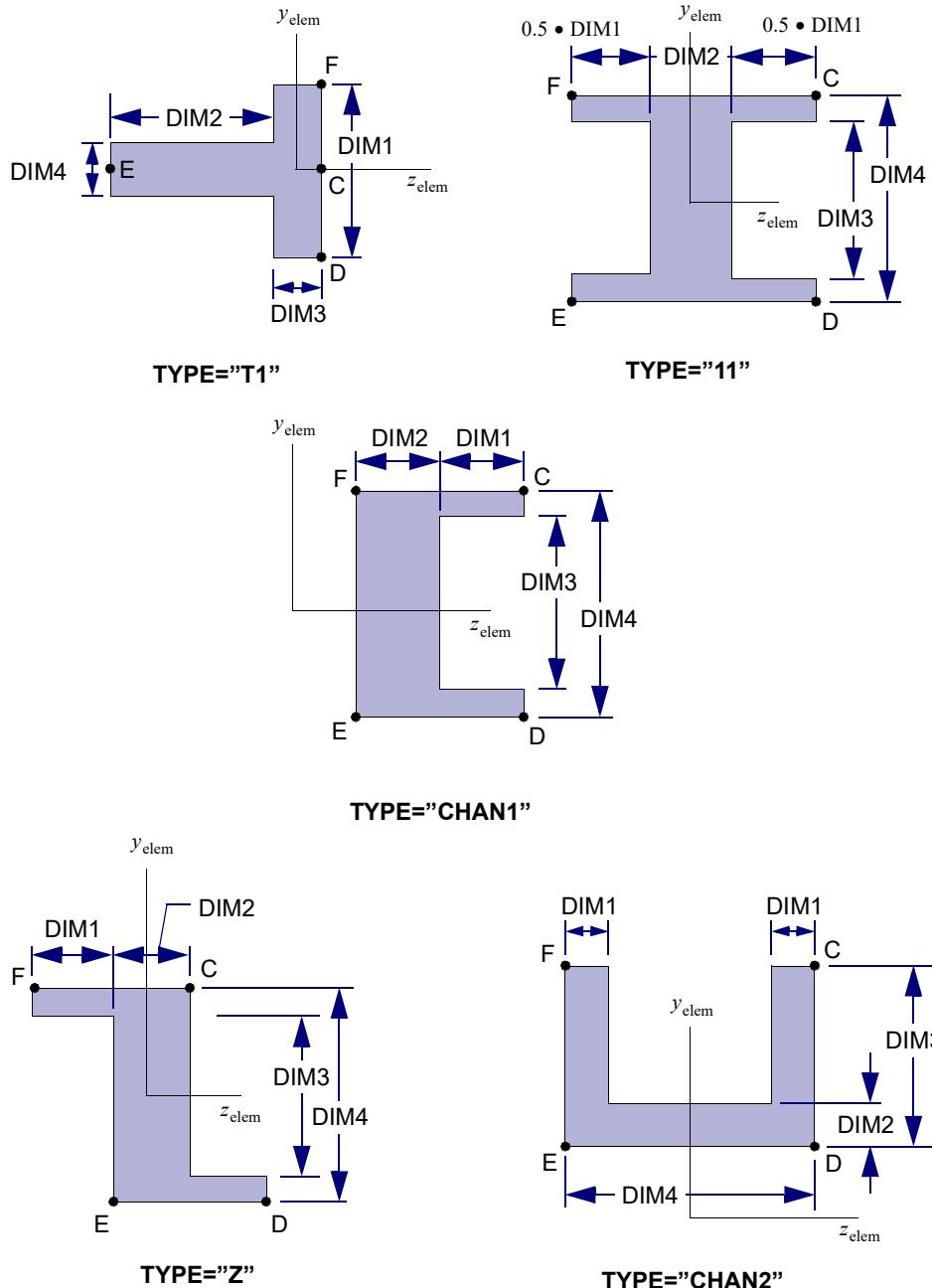


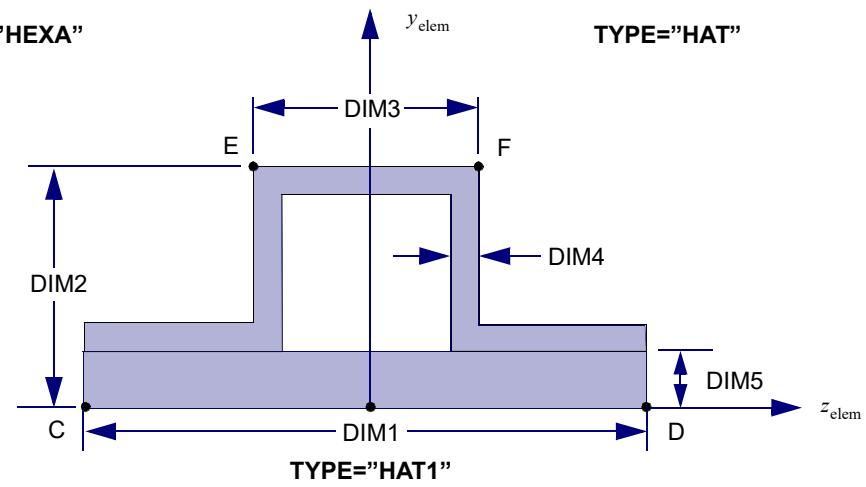
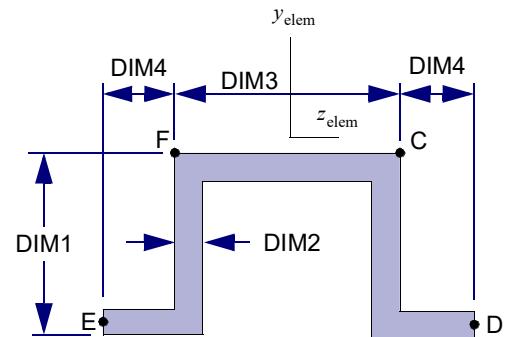
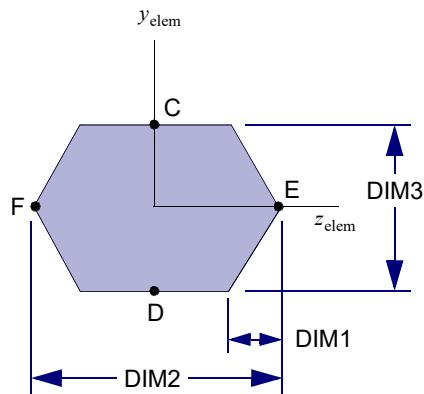
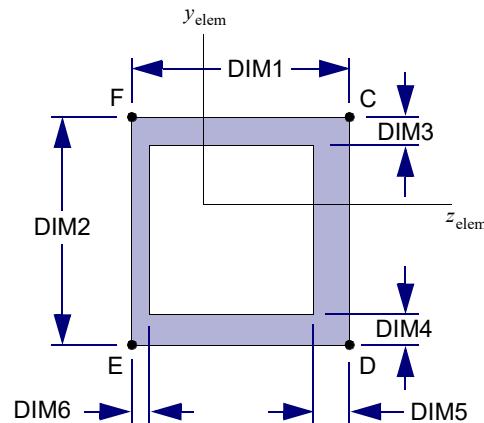
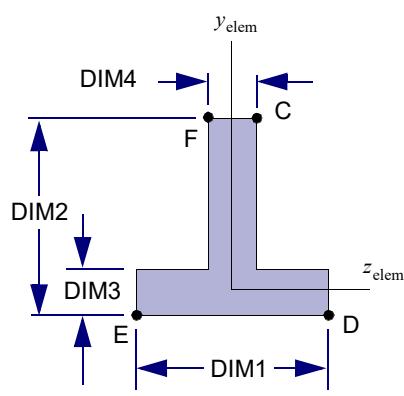
**TYPE="I"**



**TYPE="CHAN"**







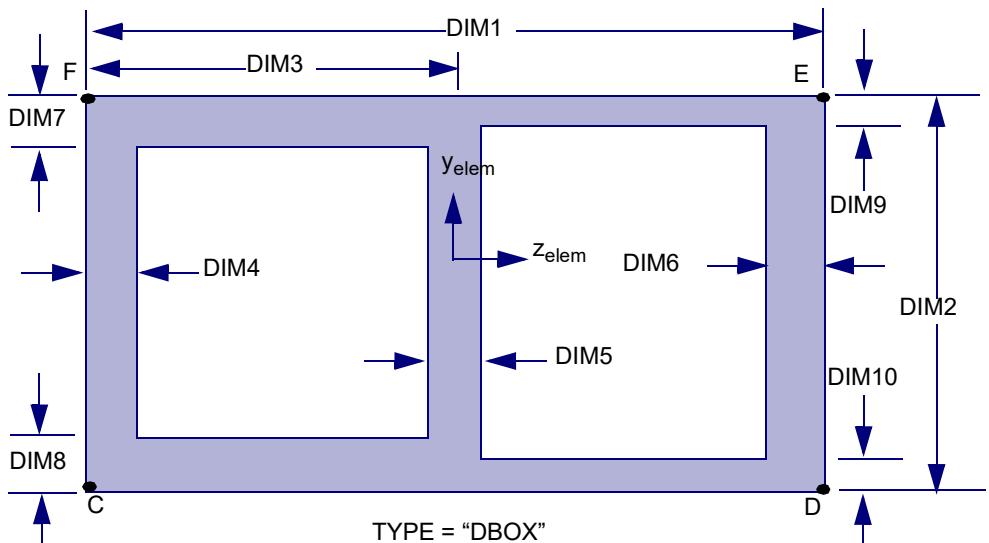


Figure 9-126 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

**PBELT****Belt Property- SOL 700**

Defines the properties of a belt element referenced by a CROD entry. Used in SOL700 only.

**Format:**

| 1     | 2   | 3    | 4      | 5       | 6     | 7     | 8     | 9         | 10 |
|-------|-----|------|--------|---------|-------|-------|-------|-----------|----|
| PBELT | PID | LOAD | UNLOAD | DENSITY | DAMP1 | DAMP2 | SLACK | PRESTRESS |    |
|       |     |      |        |         |       |       |       |           |    |

**Example:**

|       |   |    |    |        |     |     |  |  |  |
|-------|---|----|----|--------|-----|-----|--|--|--|
| PBELT | 9 | 12 | 12 | 2.0E-5 | 0.1 | 0.1 |  |  |  |
|       |   |    |    |        |     |     |  |  |  |

| Field   | Content   |
|---------|---|
| PID     | Unique belt property number. (Integer > 0; required)  |
| LOAD    | TABLED1 ID defining the force as a function of strain during loading. The strain at time n is specified as engineering strain. (Integer > 0; required)<br><br>$\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$ |
| UNLOAD  | TABLED1 ID defining the force as a function of strain during unloading. The strain at time is specified as engineering strain. (Integer > 0; required)<br><br>$\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$ |
| DENSITY | Density of the belt elements as mass per unit length. (Real > 0.0; required)  |

| Field     | Content   |
|-----------|---|
| DAMP1     | <p>A damping force is added to the internal force of the belt elements to damp out high frequency oscillations. (Real &gt; 0.0; default=0.1)</p> <p>The damping force is equal to:</p> $F_{damp} = DAMP1 \cdot mass \cdot (dvel/dt)$ <p>Where</p> $F_{damp} = \text{damping force}$ $DAMP1 = \text{damping coefficient}$ $\text{mass} = \text{mass of belt element}$ $dvel = \text{velocity of elongation}$ $dt = \text{time step}$ |
| DAMP2     | <p>The maximum damping force: <math>DAMP2 * F_{belt}</math>. (Real &gt; 0.0; default=0.1)</p> <p>Where</p> $F_{belt} = \text{Internal force in the belt element}$   |
| SLACK     | <p>TABLED1 ID defining the slack as a function of time. (Integer &gt; 0; default=not used)</p> <p>The slack must be specified as engineering strain and will be subtracted from the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) - SLACK(n)$ <p>The force in the element is zero until the element strain exceeds the slack.</p>   |
| PRESTRESS | <p>TABLED1 ID defining a prestress strain as a function of time. (Integer &gt; 0; default=not used)</p> <p>The prestress strain must be specified as engineering strain and will be added to the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) + PRESTRESS(n)$  |

**Remarks:**

1. The loading and unloading curves must start at (0.0, 0.0).

2. During loading, the loading curve is applied to determine the force in the belt element. At unloading, the unloading curve is shifted along the strain axis until it intersects the loading curve at the point from which unloading commences. The unloading table is applied for unloading and reloading, until the strain again exceeds the intersection point. Upon further loading, the loading table is applied. For subsequent unloading, the sequence is repeated.
3. Belt elements are tension only elements.
4. Instantaneous slack of an element can also be initialized per element using the TICEL entry with the keyword SLACK and a corresponding VALUE.

**PBEMN1**

## Nonlinear Property Extensions for a PBEAM or PBEAML Entry

Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry in SOL 400.

**Format:**

| 1      | 2    | 3    | 4    | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|------|------|------|------|---|---|---|---|----|
| PBEMN1 | PID  | MID  |      | SECT |   |   |   |   |    |
|        | “C2” | BEH2 | INT2 |      |   |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PBEMN1 | 27 | 93 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number of an existing PBEAM entry. (Integer > 0)  |
| MID        | Material ID. Remark 9. (Integer $\geq 0$ )  |
| SECT       | Section integration. SECT = “S” a smeared cross section is used for integration. SECT = “N” a numerically integrated cross section is used. See Remark 10. (Character Default S or blank) |
| C2         | Keyword indicating that items following apply to elements with two end grids. (Character)   |
| BEH2       | Element structural behavior. See Remark 5. (Character Default BEAM)   |
| INT2       | Integration scheme. See Remarks 5. and 6. (Character Default LC)  |

**Remarks:**

1. The PID above must point to an existing PBEAM or PBEAML Bulk Data entry and is honored only in SOL 400.
2. Tapering of the CBEAM is ignored. Only section properties at end A are used.
3. MID if blank (or 0) use the MID value on the PBEAM or PBEAML entry. If > 0 it will override the MID value on the PBEAM entry.
4. The MID entry may point to MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

| Implicit Structural Materials |
|-------------------------------|
| MAT1                          |
| MATVE                         |
| MATVP                         |

| Implicit Structural Materials |
|-------------------------------|
| MATEP                         |
| MATF                          |
| MATS1                         |

5. BEH2 refers to the nonlinear structural behavior of the BEAM element. An underlined item delineates a default.

| Structural Classification of Elements |            |                  |              |         |
|---------------------------------------|------------|------------------|--------------|---------|
| Element Structural Type               | BEHAV CODE | Integration Code | Element Type | # Nodes |
| Beam                                  | BEAM       | LC               | BEAM         | 2       |
|                                       |            | LCC              | BEAM         | 2       |
|                                       |            | LCO              | BEAM         | 2       |
|                                       |            | LS               | BEAM         | 2       |

6. Integration codes in Remark 5. are:

| INT CODE | Integration Type            |
|----------|-----------------------------|
| LC       | Linear/Cubic                |
| LCC      | Linear/Cubic Closed section |
| LCO      | Linear/Cubic Open section   |
| LS       | Linear-shear                |

7. Integration code LCO requires appropriate scalar point SA and SB entries on the CBEAM entry or a fatal message will result.
8. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.
9. The structural element damping coefficient, GE, is not supported on elements which reference PBEMN1.

10. Smeared cross section support for integration codes are as follows:

| <b>Primary BEAM Property</b> | <b>INT CODE</b> | <b>SECT</b> | <b>Interpolation</b>   | <b>Usage</b>   | <b>COMMENTS</b>   |
|------------------------------|-----------------|-------------|--|--|---|
| PBEAM                        | LC              | S           | Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements | Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam                                       | Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.   |
|                              | LS              | S           | Linear Interpolation for axial displacements, transverse displacements and rotations             | Both Thin-Walled and Solid Sections with elastic materials. Captures transverse shear effects - useful for deep beams. | Smeared section properties only are supported. Only linear elastic materials via MAT1 card can be provided.   |
|                              | LC or LS        | N           |  |  | Not Supported   |
|                              | LCC or LCO      | S           |  |  | Not Supported   |
|                              | LCC or LCO      | N           |  |  | Not Supported   |
| PBEAML                       | LC or LS        | S           |  |  | Not Supported   |
|                              | LC              | N           | Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements | Solid Sections. Euler-Bernoulli Beam   | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported |

| Primary BEAM Property | INT CODE   | SECT | Interpolation  | Usage  | COMMENTS  |
|-----------------------|------------|------|--|--|---|
|                       | LS         | N    | Linear Interpolation for axial displacements, transverse displacements and rotations             | Solid Sections. Captures transverse shear effects - useful for deep beams. | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported |
|                       | LCC or LCO | S    |  |  | Not Supported   |
|                       | LCC        | N    | Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements | Thin-Walled Closed Sections without warping. Euler-Bernoulli Beam          | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported |
|                       | LCO        | N    | Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements | Thin-Walled Open Sections with warping. Euler-Bernoulli Beam               | Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported |

**PBEND****Curved Beam or Pipe Element Property**

Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).

**Format:**

| 1     | 2   | 3   | 4   | 5  | 6  | 7      | 8  | 9      | 10 |
|-------|-----|-----|-----|----|----|--------|----|--------|----|
| PBEND | PID | MID | A   | I1 | I2 | J      | RB | THETAB |    |
|       | C1  | C2  | DI  | D2 | E1 | E2     | F1 | F2     |    |
|       | K1  | K2  | NSM | RC | ZC | DELTAN |    |        |    |

**Example:**

|       |     |     |      |      |      |      |     |  |  |
|-------|-----|-----|------|------|------|------|-----|--|--|
| PBEND | 39  | 1   | 0.8  | 0.07 | 0.04 | 0.04 | 10. |  |  |
|       | 0.5 | 0.4 | -0.5 | 0.4  |      |      |     |  |  |
|       | 0.6 | 0.6 |      |      |      | 0.1  |     |  |  |

**Alternate Format and Example for Elbows and Curved Pipes:**

|       |     |     |     |     |      |     |     |        |  |
|-------|-----|-----|-----|-----|------|-----|-----|--------|--|
| PBEND | PID | MID | FSI | RM  | T    | P   | RB  | THETAB |  |
|       |     |     | NSM | RC  | ZC   |     |     |        |  |
| PBEND | 39  | 1   | 1   | 0.5 | 0.02 | 10. | 10. |        |  |
|       |     |     |     |     | 0.1  |     |     |        |  |

| Descriptor     | Meaning   |
|----------------|---|
| PID            | Property identification number. (Integer > 0)   |
| MID            | Material identification number. See Remarks 1. and 2. (Integer > 0)                                     |
| A              | Area of the beam cross section. (Real > 0.0)  |
| I1, I2         | Area moments of inertia in planes 1 and 2. (Real > 0.0)   |
| J              | Torsional stiffness. (Real > 0.0)   |
| FSI            | Flag selecting the flexibility and stress intensification factors. See Remark 3. (Integer = 1, 2, or 3) |
| RM             | Mean cross-sectional radius of the curved pipe. (Real > 0.0)  |
| T              | Wall thickness of the curved pipe. (Real $\geq$ 0.0; RM + T/2 < RB)                                     |
| P              | Internal pressure. (Real)   |
| RB             | Bend radius of the line of centroids. (Real. Optional, see CBEND entry.)                                |
| THETAB         | Arc angle of element. (Real, in degrees. Optional, see CBEND entry.)                                    |
| Ci, Di, Ei, Fi | The r,z locations from the geometric centroid for stress data recovery. See Remark 8. (Real)            |

| Descriptor | Meaning   |
|------------|---|
| K1, K2     | Shear stiffness factor K in K*A*G for plane 1 and plane 2. (Real)   |
| NSM        | Nonstructural mass per unit length. (Real)  |
| RC         | Radial offset of the geometric centroid from points GA and GB. See <a href="#">Figure 9-128</a> . (Real)  |
| ZC         | Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. See <a href="#">Figure 9-128</a> . See Remark 9. (Real)                                  |
| DELTAN     | Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature. See <a href="#">Figure 9-128</a> . See Remark 9. (Real; Default is described in Remark 5.) |

**Remarks:**

1. For structural problems, MID must reference a MAT1 material entry.
2. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
3. When the alternate PBEND entry with FSI is used the following options are available:  
Using the notation of Dodge and Moore from ORNL-TM-3658, 1972 we define:

The bend characteristic parameter  $\lambda$  as:

$$\lambda = \frac{R_B T}{R_M^2 \sqrt{(1 - \nu^2)}}.$$

The internal loading parameter  $\psi$  as

$$\psi = \frac{P R_B^2}{E R_M T}.$$

And the radius ratio parameter  $\gamma$  as

$$\gamma = \frac{R_B}{R_M}.$$

For  $T \neq 0$

$$A = 2\pi R_M T$$

$$I_1 = I_2 = \frac{1}{2} A (R_M^2 + \frac{1}{4} T^2)$$

For  $T = 0$

$$A = \pi R_M^2$$

$$I_1 = I_2 = \frac{1}{4} A R_M^2$$

For  $T \neq 0$

$$\Delta N = R_B \left\{ 1 - \frac{1}{2} \left[ \sqrt{(1 - \gamma^{-2} - \frac{1}{4} (\frac{T}{R_B})^2 + (\frac{T}{R_B}) \gamma^{-1})} + \sqrt{(1 - \gamma^{-2} - \frac{1}{4} (\frac{T}{R_B})^2 - (\frac{T}{R_B}) \gamma^{-1})} \right] \right\}.$$

For  $T = 0$

$$\Delta N = \frac{1}{2} R_B [1 - \sqrt{(1 - \gamma^{-2})}]$$

### Element flexibility calculations:

The shear stiffness factor represents a correction to the section area by computing an effective area  $A_s = KA$ . Here K is defined as:

$$K = \frac{1}{f_s}$$

$f_s$  is the form factor obtained here from the principle of complementary virtual work. There are many other ways to obtain this factor. By complementary virtual work it is defined as:

$$f_s = \frac{A}{I_1^2} \int \frac{Q^2}{b^2} dA,$$

where Q is the first moment of area and b is the width of section cut.

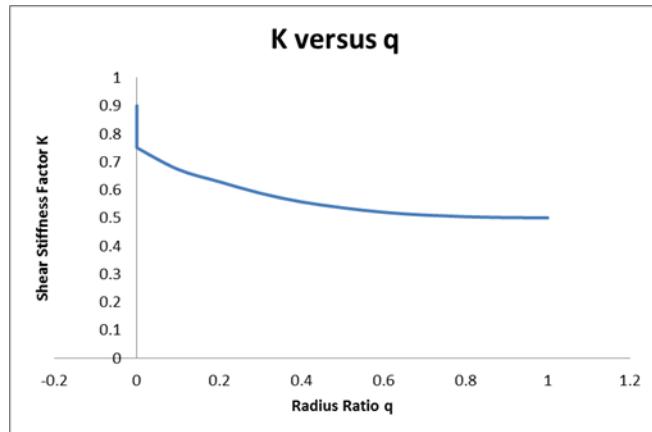
For a solid circular cross-section the above expression yields a value of  $K = 0.9$ , and for very thin walled pipes the above integral evaluates as  $\pi T R_m^5$  and we get a value of  $K = 0.5$ .

For pipes with general thickness  $T > 0$  we define the radius ratio as:

$$q = \frac{R_i}{R_o} = \frac{1 - \frac{1}{2} \frac{T}{R_M}}{1 + \frac{1}{2} \frac{T}{R_M}}.$$

where  $T = R_o - R_i$  and  $R_o$  is the outer radius of the pipe and  $R_i$  is the inner radius of the pipe. The above integral becomes a complicated function in terms of radicals and arc sine functions of q and its explicit integration form is not warranted in view of the approximate nature for  $f_s$ . We therefore approximate with the curve fit

$$K_1 = K_2 = \frac{0.75}{1 + \frac{q}{1+q^2}}$$



The in and out of plane flexibility factors  $K_p$  used in computing the element flexibility are calculated as:

For FSI = 1

$$K_p = 1.0$$

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

$$K_p = \frac{1.65 R_M^2}{R_B T} \left[ \frac{1}{1 + 6 \frac{P R_M}{T E} \left( \frac{R_M}{T} \right)^{\frac{4}{3}} \gamma^{\frac{1}{3}}} \right]$$

where it is recommended that  $\lambda \geq 0.2$

For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$K_p = \frac{1.73}{\lambda} \left[ \frac{1}{1 + 1.75 \lambda^{-\frac{4}{3}} \exp(-1.15 \psi^{-\frac{1}{4}})} \right]$$

where it is recommended that  $0.05 \leq \lambda \leq 1.0$  and  $0.0 \leq \psi \leq 0.1$ . (Note in the cited Dodge and Moore the  $K_p$  coefficient has a value of 1.66 which was updated in 1991 to the current value of 1.73)

Also we should have  $K_p \geq 1.0$  for all FSI.

#### Stress recovery stress intensification factor calculations:

Define  $S_1$  as the in plane stress intensification factor and  $S_2$  as the out of plane stress intensification factor.

For FSI = 1

$$S_1 = \frac{I_1}{AR_B} \left[ \frac{1}{r_i} + \frac{R_B - \Delta N}{\Delta N(R_B + r_i)} \right]$$

$$S_2 = 1.0$$

If  $\Delta N = 0.0$ ,  $T = 0.0$ ,  $R_M = 0.0$  then  $S_1 = 1.0$ .

$r_i$  is the C1, D1, E1, F1 of the (r,z) recovery locations.

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

Define:

$$X_1 = 5.0 + 6\lambda^2 + 24\psi$$

$$X_2 = 17.0 + 600\lambda^2 + 480\psi$$

$$X_3 = X_1 X_2 - 6.25$$

$$X_4 = (1 - \nu^2)(X_3 - 4.5X_2)$$

and

$$S_{1t} = \sin \phi + [(1.5X_2 - 18.75) \sin 3\phi + 11.25 \sin 5\phi] / X_4$$

$$S_{1n} = \nu \lambda (9X_2 \cos 2\phi + 225 \cos 4\phi) / X_4$$

$$S_1 = S_{1t} + S_{1n}$$

$$S_{2t} = \cos \phi + [(1.5X_2 - 18.75) \cos 3\phi + 11.25 \cos 5\phi] / X_4$$

$$S_{2n} = \nu \lambda (9X_2 \sin 2\phi + 225 \sin 4\phi) / X_4$$

$$S_2 = S_{2t} + S_{2n}$$

For data recovery the circumferential angle  $\phi$  is evaluated at 0 deg, 90 deg, 180 deg, and 270 deg. For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$S_1 = S_2 = \frac{2\lambda^{-\frac{2}{3}}(1.0 + 0.25\gamma^{-1})}{1.0 + \lambda^{-\frac{4}{3}} \exp(-\psi^{-\frac{1}{4}})}.$$

4. The transverse shear stiffness in planes 1 and 2 are  $K1*A^*G$  and  $K2*A^*G$ , respectively. The default values for K1 and K2 on the first format are zero, which means the transverse shear flexibilities ( $1/Ki*A^*G$ ) are set equal to zero. Transverse shear stiffness for the alternate format are automatically calculated for the curved pipe.
5. The neutral axis radial offset from the geometric centroid is default to the

$$\Delta N = \frac{II}{A \cdot RB}$$

It is recommended that the default be used whenever

$$\frac{(RB)^2 A}{II} < 15$$

in which case the default value of  $\Delta N$  is within 5% of the exact expression for circular or rectangular cross sections. For the alternate format, the neutral axis offset is calculated from an analytical expression for a hollow or solid circular cross section.

The user may compute an exact value for  $N$  as follows:

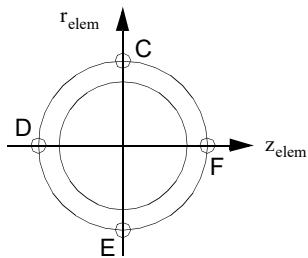
$$\Delta N = \frac{RB}{1 + \frac{(RB)^2 A}{Z}}$$

where

$$Z = \int \frac{r^2 dA}{1 + \frac{r}{RB}}$$

The integration is carried out over the cross section of the element.

6. If T is zero, a solid circular cross section of radius RM is assumed and FSI must be 1.
7. If the first format is used, third-order moments are neglected for the consistent mass matrix. These moments are zero whenever the cross section of the beam is symmetric about both the r and z axes.
8. If the circular cross-sectional property entry format is used, the stress points are automatically located at the points indicated in [Figure 9-127](#).
9. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
  - Offset vectors are not affected by thermal loads.
  - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200; SOLs 101,103 and 107 through 112 with STATSUB; and also nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)



[Figure 9-127 PBEND Circular Cross Section](#)

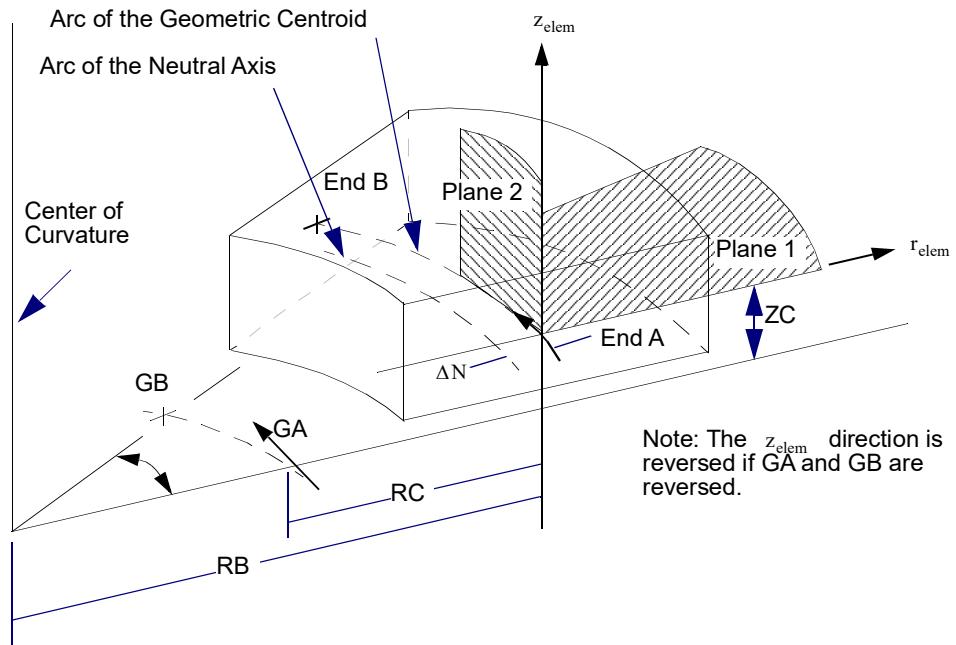


Figure 9-128 PBEND Element Coordinate System

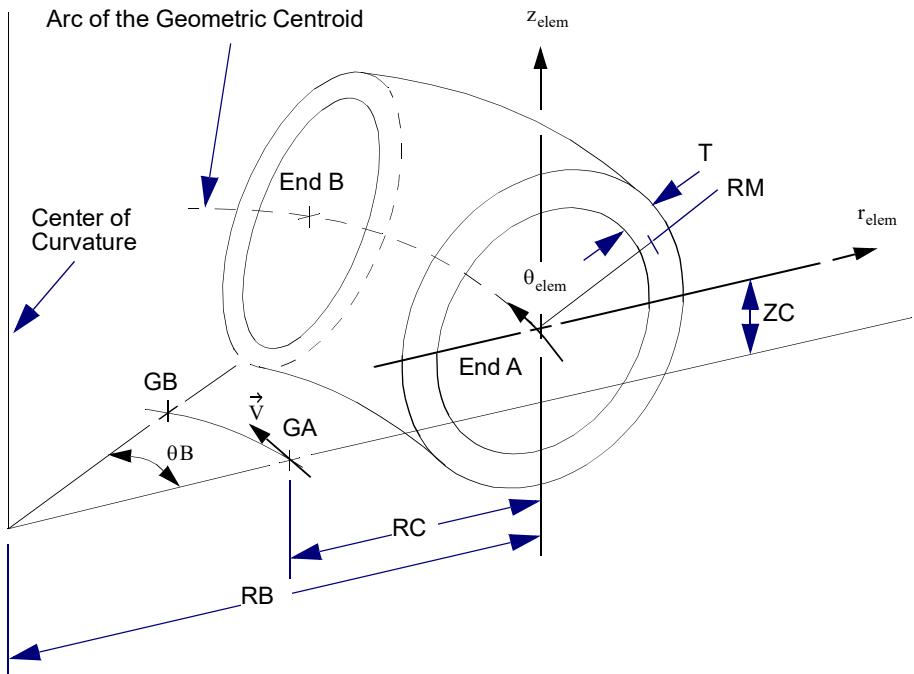


Figure 9-129 PBEND Circular Cross Section Element Coordinate System

10. For RC network solver in thermal analysis, the DELTAN is ignored.
11. PBEND is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEND property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEND entries.

**PBMARB6****Arbitrary Beam/Bar Cross Section in SOL 600**

Defines arbitrary BEAM/BAR cross section for use in SOL 600.

**Format:**

| 1       | 2   | 3   | 4    | 5    | 6    | 7      | 8    | 9    | 10      |
|---------|-----|-----|------|------|------|--------|------|------|---------|
| PBMARB6 | PID | MID | Nseg | ORDx | ORDy | PreInt | POIS | IPRN |         |
|         | Px  | Py  | F1   | F2   | F3   | F4     | F5   | F6   |         |
|         | X1  | Y1  | X2   | Y2   | X3   | Y3     | X4   | Y4   | 1st seg |
|         | X1  | Y1  | X2   | Y2   | X3   | Y3     | X4   | Y4   | 2nd seg |

(more rectangular segments, if required)

|  |    |    |    |    |    |    |    |    |          |
|--|----|----|----|----|----|----|----|----|----------|
|  | X1 | Y1 | X2 | Y2 | X3 | Y3 | X4 | Y4 | last seg |
|--|----|----|----|----|----|----|----|----|----------|

**Example: (Remark 5)**

|         |     |     |     |     |     |     |     |     |  |
|---------|-----|-----|-----|-----|-----|-----|-----|-----|--|
| PBMARB6 | 20  | 4   | 2   | 3   | 3   |     |     | 3   |  |
|         | 2.0 | 1.0 |     |     |     |     |     |     |  |
|         | 0.0 | 1.0 | 0.8 | 0.0 | 0.8 | 1.0 | 0.0 | 1.0 |  |
|         | 0.0 | 1.0 | 1.0 | 1.0 | 1.0 | 2.0 | 0.0 | 2.0 |  |

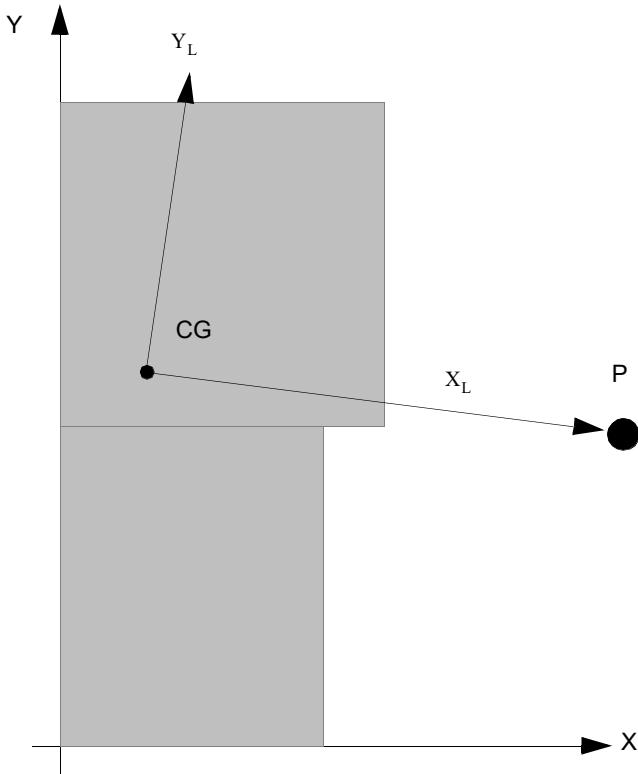
| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. Must be unique among all property ID's. (Integer > 0; no Default)   |
| MID        | Material identification number. (Integer > 0; no Default)   |
| Nseg       | Number of quadrilateral shaped segments. See Remark 2. (Integer; no Default)  |
| ORDx       | Order of the integration rule used for each quadrilateral shaped segment in parametric - direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default)   |
| ORDy       | Order of the integration rule used for each quadrilateral shaped segment in parametric - direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is the same rule and order as in parametric -direction and the order cannot be larger than 10. (Integer; no Default) |
| PreInt     | Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)  |
| POIS       | For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)   |

| Descriptor | Meaning   |
|------------|---|
| IPRN       | <p>Enter 1 to have the principal axis associated with the largest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 2 to have the principal axis associated with the smallest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 3 to have the x-axis of the coordinate system for which the section is being defined to be aligned with the local x-axis of the element (element coordinates). (Integer; Default = 3)</p>   |
| Px         | Enter the x-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the x-coordinate of a point on the positive local x-axis. This coordinate defaults to $X_{cg}+1$ , where $X_{cg}$ is the x-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity. |
| Py         | Enter the y-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the y-coordinate of a point on the positive local x-axis. This coordinate defaults to $Y_{cg}$ , where $Y_{cg}$ is the y-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity.   |
|            | F1 Normal stiffness factor F1. (Real; Default = 1.0)  |
|            | F2 Bending stiffness factor for bending about local X axis. (Real; Default = 1.0)   |
|            | F3 Bending stiffness factor for bending about local Y Axis. (Real; Default = 1.0)   |
|            | F4 Shear stiffness factor for shear in local X direction. (Real; Default = 1.0)   |
|            | F5 Shear stiffness factor for shear in local Y direction. (Real; Default = 1.0)   |
|            | F6 Torsional stiffness factor. (Real; Default = 1.0)  |
|            | $X_i, Y_i$ Coordinates of the 4 corners of each rectangle. Enter Nseg sets of 4 coordinates each. See Remark 3. (Real; no Default)  |

### Remarks:

1. This option corresponds to Marc's BEAM SECT Method D.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. The corners are given in counterclockwise order with respect to the local x-y axis.
4. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.

5. For the example, a section is made up of two rectangles as shown in the figure below. The input for the section defines one section with two quadrilateral segments using a 3x3 Simpson integration. The lower segment has size 0.8x1 and the upper segment has size 1x1. The first principal axis defines the local x-axis and the vector from the center of gravity (CG) to the projection of point P (2.0,1.0) onto this principal axis defines its positive direction. The coordinates of corner points of the segments and the coordinates of the point P have been entered with respect to the x,y-system.



6. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.  
 7. If this entry is used, the following parameters must be included in the bulk data:

PARAM,MARCBEAM,0

PARAM,MROUTLAY,N      (where N is the number of integration layers, suggested values are N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N      (N should be the same as specified for MROUTLAY)

**PBMNUM6**

## Four Specific Numerically Integrated BEAM/BAR Cross Section in SOL 600

Defines four specific numerically integrated BEAM/BAR cross section for use in SOL 600.

**Format:**

| 1       | 2   | 3   | 4     | 5    | 6    | 7      | 8    | 9 | 10 |
|---------|-----|-----|-------|------|------|--------|------|---|----|
| PBMNUM6 | PID | MID | ITYPE | ORDx | ORDy | PreInt | POIS |   |    |
|         | A   | B   | C     |      |      |        |      |   |    |
|         | F1  | F2  | F3    | F4   | F5   | F6     |      |   |    |

**Example:**

|         |     |     |   |   |   |  |  |  |  |
|---------|-----|-----|---|---|---|--|--|--|--|
| PBMNUM6 | 30  | 6   | 2 | 3 | 3 |  |  |  |  |
|         |     |     |   |   |   |  |  |  |  |
|         | 7.0 | 2.5 |   |   |   |  |  |  |  |

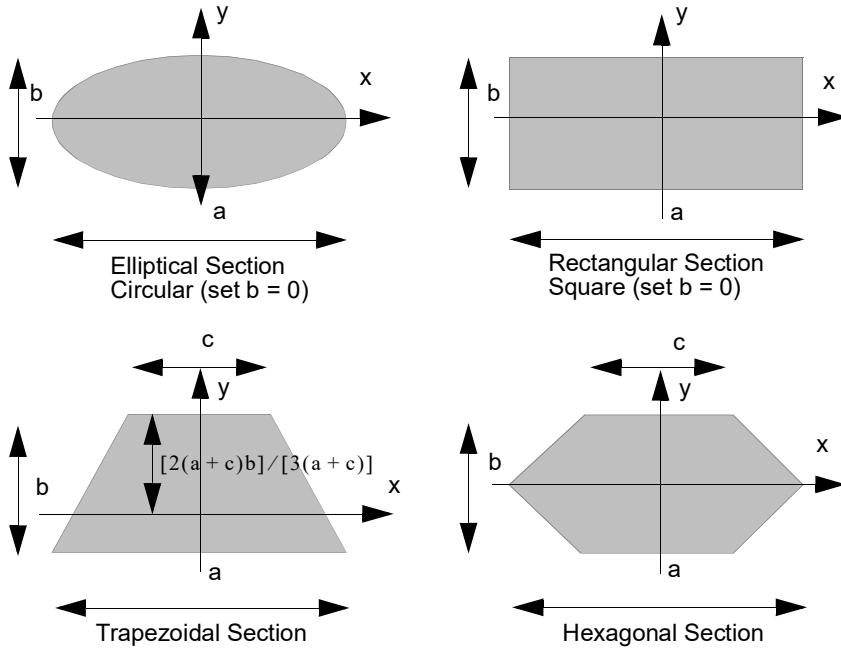
| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0; no Default) Must be unique among all property ID's   |
| MID        | Material identification number. (Integer > 0; no Default)  |
| ITYPE      | Type of section, see figure below. (Integer; no Default) <ul style="list-style-type: none"> <li>1      Elliptical</li> <li>2      Rectangular</li> <li>3      Trapezoidal</li> <li>4      Hexagonal</li> </ul>   |
| ORDx       | For an elliptical section, enter the number of subdivisions in radial direction. The default is 3.<br><br>For a rectangular or trapezoidal section, enter the order of the integration rule used in local x-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10.<br><br>For a hexagonal section, enter the order of the integration rule used in local x-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default) |

| Descriptor | Meaning   |
|------------|---|
| ORDy       | For an elliptical section, enter the number of subdivisions in circumferential direction of a 90° sector. The default is 2. For a rectangular or trapezoidal section, enter the order of the integration rule used in local y-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is what was entered in the first field and the order cannot be larger than 10.  |
|            | For a hexagonal section, enter the order of the integration rule used in local y-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton- Cotes rule is used. If the number is negative, a Gauss rule is used. The default is half the order in x-direction plus 1 and the order cannot be larger than 10. For non-Gauss rules, the points in the top row of the lower trapezoid coinciding with the points in the bottom row of the upper trapezoid (i.e., the points coinciding at $y = 0$ ) are merged together. |
| PreInt     | Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)  |
| POIS       | For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)   |
| A          | First dimension of the cross section (Real; no Default) <ul style="list-style-type: none"> <li>Ellipse A is the diameter of the circle or the length of the ellipse in local x.</li> <li>Rectangle A is the length of the square or the rectangle in local x.</li> <li>Trapezoid A is the width of the trapezoid in local x on minus local y side.</li> <li>Hexagon A is the width of the hexagon in local x at <math>y = 0</math>.</li> </ul>  |
| B          | Second dimension of the cross section (Real; Default B=A) <ul style="list-style-type: none"> <li>Ellipse B is the height of the ellipse in local y.</li> <li>Rectangle B is the height of the rectangle in local y.</li> <li>Trapezoid B is the height of the trapezoid in local y.</li> <li>Hexagon B is the height of the hexagon in local y.</li> </ul>  |
| C          | Third dimension of the cross section if required (Real; Default = 0.0) <ul style="list-style-type: none"> <li>Ellipse C is not used. Leave blank or enter 0.</li> <li>Rectangle C is not used. Leave blank or enter 0.</li> <li>Trapezoid C is the width of the trapezoid in local x on the plus local y side.</li> <li>Hexagon C is the width of the hexagon in local x on either local y side.</li> </ul>   |
| F1         | Normal stiffness factor F1 (Real; Default = 1.0)  |
| F2         | Bending stiffness factor for bending about local X axis (Real; Default = 1.0)   |
| F3         | Bending stiffness factor for bending about local Y Axis (Real; Default = 1.0)   |
| F4         | Shear stiffness factor for shear in local X direction (Real; Default = 1.0)   |

| Descriptor | Meaning   |
|------------|---|
| F5         | Shear stiffness factor for shear in local Y direction (Real; Default = 1.0) |
| F6         | Torsional stiffness factor. (Real; Default = 1.0)                           |

**Remarks:**

1. This option corresponds to Marc's BEAM SECT Method C.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.
4. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.
5. The second continuation entry may be eliminated if F1 through F6 take the default values.
6. For the rectangle, if B=A or B=0.0 the cross section is a square.
7. For the Ellipse, if B=A or B=0.0, the cross section is a circle.



8. If this entry is used, the following parameters must be included in the bulk data:

PARAM,MARCBEAM,0

PARAM,MROUTLAY,N

(where N is the number of integration layers, suggested values are  
N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N

(N should be the same as specified for MROUTLAY)

## PBMSECT

## Arbitrary Cross-Section for CBEAM

Defines the shape of arbitrary cross-section for CBEAM element.

### Format:

| 1                                      | 2   | 3   | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|--|-----|-----|------|---|---|---|---|---|----|
| PBMSECT                                | PID | MID | FORM |   |   |   |   |   |    |
| Data description for arbitrary section |     |     |      |   |   |   |   |   |    |

### Example:

|   |   |    |    |  |  |  |  |  |  |
|---|---|----|----|--|--|--|--|--|--|
| PBMSECT   | 1 | 10 | GS |  |  |  |  |  |  |
| OUTP=10,INP=20  |   |    |    |  |  |  |  |  |  |
| PBMSECT   | 1 | 10 | CP |  |  |  |  |  |  |
| OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)], NSM=0.01 |   |    |    |  |  |  |  |  |  |

| Descriptor | Meaning                                       |
|------------|---|
| PID        | Property identification number. (Integer > 0) |
| MID        | Material identification number. (Integer > 0) |
| FORM       | Cross-section form. (Character) See Remark 1. |

### Remarks:

- Options for FORM are

|    |                 |
|----|-----------------|
| GS | General Section |
| OP | Open Profile    |
| CP | Closed Profile  |

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

- OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.
- OUTM = value(Integer > 0), points to the ID of BEGIN BULK ARBMODEL. OUTM is designed specifically for arbitrary beam cross section with finite element discritization already available. Note that OUTM must not appear together with other keyword, such as OUTP or INP, on a PBMSECT.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch. The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

T(id) = [value(Real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the start and end points of line segment(s). The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=() do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

- Core(id) = [PCID,PT=(pid1,pid2)]; specifies the composite layup for CORE part of composite. PCID is the ID of a PCOMPi/PCOMPG Bulk Data entry. PT=(pid1,pid2) defines the start and end points of line segment(s) which utilizes PCID.
- Core=PCID is acceptable input and is used as default which is applicable to all segments that are not specifically defined with Core(id).
  - Thickness continuity of a ply must be maintained. Thickness change from segment to segment is not allowed.
  - For CP, the closing segment should always use the default Core=pcid.
  - No ply should go over the profile line which is defined by OUTP and BRP. If situation arises, split a ply into two in such a way the no ply go over the profile line.
- Layer(id) = [PCID,SETID]; specifies the composite layup for additional Layer(s) that wraps around Core. PCID is the ID of a PCOMP/PCOMPG Bulk Data entry. SETID selects a SET1/SET3 with POINT IDs.
- NSM = value(Real > 0.0), specifies non-structural mass per unit length.
- (id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBMSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.
3. Stress data recovery points are selected automatically from all points of a PBMSECT with GS form. The points with maximum and/or minimum coordinates in X and Y axes are more likely to be picked. For PBMSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
  4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.

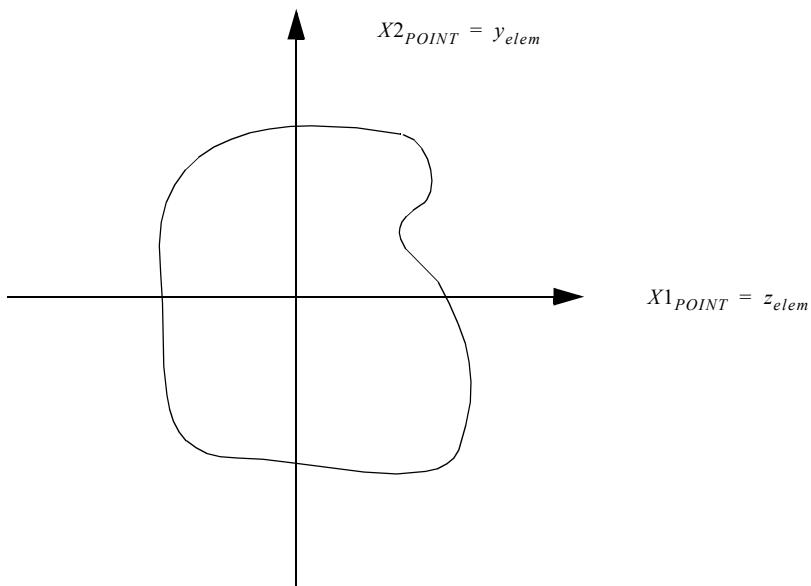


Figure 9-130 Arbitrary Cross-Section Definition

5. Current implementation of PBMSECT supports constant section beam only.
6. Note that keyword Core can be abbreviated as 'C'. Similarly, keyword Layer can be abbreviated as 'L'.
7. If Core and/or Layer appears in PBMSECT, the PBMSECT ID can not be referenced on CBEAM. Instead, it should be referenced on CBEAM3. Note, however, that a CBEAM3 cannot be used with any other of the above options except the Core and/or Layer option. The use of any other option results in a FATAL. In addition, it is recommended that CBEAM3 referencing composite PBMSECT has 3 nodes and 3 warping DOFs.'
8. If OUTM=arbid is utilized on PBMSECT, element connection, grid location, PSHELL and material Bulk Data entries must be provided after 'BEGIN ARBMODEL=arbid'.
9. Note that the 'arbid' used under 'BEGIN ARBMODEL' is considered global and can be referenced by PBMSECT with OUTM=arbid in different 'BEGIN SUPER' Bulk Data Section for Part Superelements (SE).
10. PBMSECT with Core or Layer must be utilized along with 'PARAM,ARBMSTYP,TIMOSHEN' in the Bulk Data Section.
11. The entry computes, based on an internally generated finite element analysis using a 2D mesh of the cross-section, the following:

$$A = \int dy dz$$

$$I_1 = \int y^2 dy dz$$

$$I_2 = \int z^2 dy dz$$

$$I_{12} = \int yz dy dz$$

where the above integrals are evaluated by numerical integration.

For a beam cross-section, the warping function,  $\phi$ , satisfies the equation

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

with boundary

$$\left( \frac{\partial \phi}{\partial y} + z \right) n_y + \left( \frac{\partial \phi}{\partial z} - y \right) n_z = 0$$

where  $n_y$  and  $n_z$  are the direction cosine of the normal to the boundary.

Then, the torsion constant is defined as

$$J = I_1 + I_2 - \int \left[ \frac{\partial \phi}{\partial z} \frac{\partial \phi}{\partial y} \right] \begin{Bmatrix} -y \\ z \end{Bmatrix} dA$$

The load equilibrium of the beam cross-section can be resolved into two Poisson equations for the shear forces in the  $y$  and  $z$  direction as:

$$\nabla^2 f_y = -y/I_1$$

$$\nabla^2 f_z = -z/I_2$$

then, the shear stiffness factor is defined as

$$K_1 = \left[ \frac{A}{I_2} \int z f_z dA \right]^{-1}$$

$$K_2 = \left[ \frac{A}{I_1} \int y f_y dA \right]^{-1}$$

The warping constant is defined as

$$C_w = \int \phi^2 dA - \left[ y_{sc} z_{sc} \right] [I] \begin{Bmatrix} y_{sc} \\ z_{sc} \end{Bmatrix}$$

The shear center is defined as

$$Q_1 = \int z dA \quad Q_2 = \int y dA$$

$$N_1 A = Q_1 / A \quad N_2 A = Q_2 / A$$

12. PBMSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBMSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

**PBRSECT****Arbitrary Cross-Section for CBAR**

Defines the shape of arbitrary cross-section for CBAR element.

**Format:**

| 1                                      | 2   | 3   | 4    | 5   | 6 | 7 | 8 | 9 | 10 |
|--|-----|-----|------|-----|---|---|---|---|----|
| PBRSECT                                | PID | MID | FORM | NSM |   |   |   |   |    |
| Data description for arbitrary section |     |     |      |     |   |   |   |   |    |

**Example:**

|   |   |    |    |  |  |  |  |  |  |
|---|---|----|----|--|--|--|--|--|--|
| PBRSECT                                       | 1 | 10 | GS |  |  |  |  |  |  |
| OUTP=10,INP=20                                |   |    |    |  |  |  |  |  |  |
| PBRSECT                                       | 1 | 10 | CP |  |  |  |  |  |  |
| OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)] |   |    |    |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0)                          |
| MID        | Material identification number. (Integer > 0)                          |
| FORM       | Cross-section form. (Character) See Remark 1.                          |
| NSM        | Non-structural mass per unit length. (Real $\geq 0.0$ ; Default = 0.0) |

**Remarks:**

- Options for FORM are

|    |                 |
|----|-----------------|
| GS | General Section |
| OP | Open Profile    |
| CP | Closed Profile  |

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

**BRP(id)** = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch.  
The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

**T(id)** = [value(real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the end points of a straight line segment. The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=( ) do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

**NSM** = value(Real > 0.0), specifies non-structural mass per unit length.  
**(id)** = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBRSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBRSECT with GS form. The points with maximum and/or minimum coordinates in X1 and/or X2 axes are more likely to be picked. For PBRSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.
5. See Remark 11. of the PBMSECT entry for the theory used to compute the cross-sectional properties.

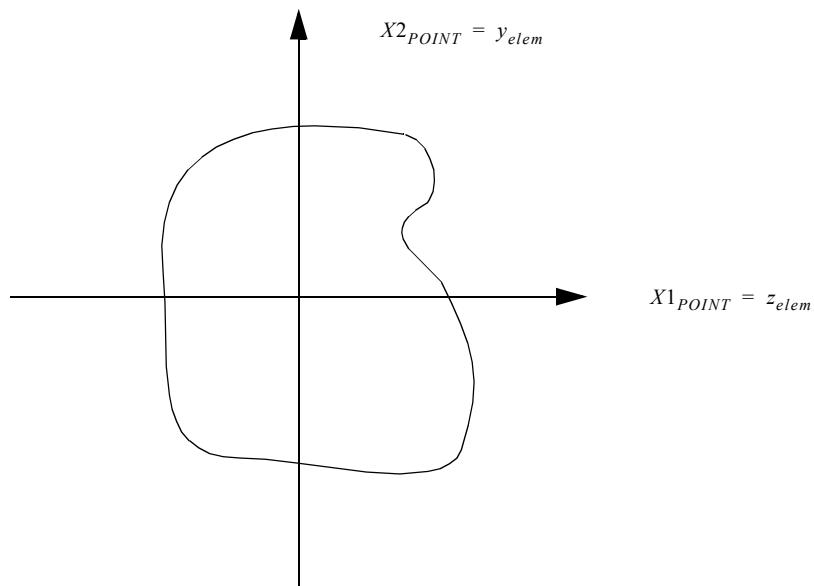


Figure 9-131 Arbitrary Cross-Section Definition

6. PBRSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBRSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

**PBSH2DT**

## 2-D Linear/Nonlinear Connection Properties using Table Lookup

Defines frequency-dependent properties of a two-dimensional element (CBUSH2D) using TABLED1 lookup.

**Format:**

| 1       | 2   | 3   | 4   | 5   | 6   | 7   | 8 | 9 | 10 |
|---------|-----|-----|-----|-----|-----|-----|---|---|----|
| PBSH2DT | PID | "K" | K11 | K22 | K12 | K21 |   |   |    |
|         |     | "B" | B11 | B22 | B12 | B21 |   |   |    |
|         |     | "M" | M11 | M22 | M12 | M21 |   |   |    |
|         |     |     |     |     |     |     |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property id of a PBUSH2D entry (see Remark 1.)   |
| K11        | TABLED1 used to define frequency-dependent stiffness in the T1 direction (Integer >0 or blank) |
| K22        | TABLED1 used to define frequency-dependent stiffness in the T2 direction (Integer >0 or blank) |
| K12        | TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)      |
| K21        | TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)      |
| B11        | TABLED1 used to define frequency-dependent damping in the T1 direction (Integer >0 or blank)   |
| B22        | TABLED1 used to define frequency-dependent damping in the T2 direction (Integer >0 or blank)   |
| B12        | TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)        |
| B21        | TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)        |
| M11        | TABLED1 used to define frequency-dependent mass in the T1 direction (Integer >0 or blank)      |
| M22        | TABLED1 used to define frequency-dependent mass in the T2 direction (Integer >0 or blank)      |
| M12        | TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)           |
| M21        | TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)           |

**Remarks:**

1. A PBUSH2D with the same PID must exist. The values from the PBUSH2D will be used as the nominal stiffness, damping, and mass. Linear statics, normal modes, reduction, and superelement processing will use the nominal values. The values from the selected TABLED1 entries will be used in any frequency-dependent loop.
2. Any field left blank indicates that the associated stiffness, damping, or mass is not frequency-dependent and the nominal values will be used for that term in the solution.

**PBUSH****Generalized Spring-and-Damper Property**

Defines the nominal property values for a generalized spring-and-damper structural element.

**Format:**

| 1     | 2   | 3     | 4     | 5    | 6     | 7   | 8   | 9   | 10 |
|-------|-----|-------|-------|------|-------|-----|-----|-----|----|
| PBUSH | PID | "K"   | K1    | K2   | K3    | K4  | K5  | K6  |    |
|       |     | "B"   | B1    | B2   | B3    | B4  | B5  | B6  |    |
|       |     | "GE"  | GE1   | GE2  | GE3   | GE4 | GE5 | GE6 |    |
|       |     | "RCV" | SA    | ST   | EA    | ET  |     |     |    |
|       |     | "M"   | M     |      |       |     |     |     |    |
|       |     | "T"   | ALPHA | TREF | COINL |     |     |     |    |

**Example 1:**

Stiffness and structural damping are specified.

|       |    |     |      |     |  |  |  |     |  |
|-------|----|-----|------|-----|--|--|--|-----|--|
| PBUSH | 35 | K   | 4.35 | 2.4 |  |  |  | 3.1 |  |
|       |    | GE  | .06  |     |  |  |  |     |  |
|       |    | RCV | 7.3  | 3.3 |  |  |  |     |  |

**Example 2:**

Damping force per unit velocity are specified.

|       |    |   |     |  |  |  |  |  |  |
|-------|----|---|-----|--|--|--|--|--|--|
| PBUSH | 35 | B | 2.3 |  |  |  |  |  |  |
|-------|----|---|-----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| "K"        | Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)  |
| Ki         | Nominal stiffness values in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)   |
| "B"        | Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)   |
| Bi         | Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks 2., 3., and 9. (Real; Default = 0.0)                   |
| "GE"       | Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)   |
| GEi        | Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. and 9. (Real; GE1 default = 0.0, GE2-6, defaults described in Remark 9.) |

| Descriptor | Meaning  |
|------------|--|
| "RCV"      | Flag indicating that the next 1 to 4 fields are stress or strain coefficients. See Remarks 4. and 5.(Character).                                     |
| SA         | Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)  |
| ST         | Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)   |
| EA         | Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)  |
| ET         | Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)   |
| "M"        | Flag indicating that the following entries are mass properties for the CBUSH element. If inertia properties ( Iij )are desired CONM2 should be used. |
| M          | Lumped mass of the CBUSH. (Real $\geq$ 0.0; Default=0.0)   |
| "T"        | Flag indicating that the following entries are thermal properties for the CBUSH element. See Remark 15.. (Character)                                 |
| ALPHA      | Thermal expansion coefficient for the CBUSH. (Real; Default=0.0)   |
| TREF       | Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 15.)  |
| COINL      | Length of a CBUSH with coincident grids. (Real; Default=0.0, COINL $\geq$ 0.0)   |

### Remarks:

1. Ki, Bi, or GEi may be made frequency dependent for both direct and modal frequency response by use of the PBUSHT entry.
2. The nominal values are used for all analysis types except frequency response and nonlinear analysis. For modal frequency response, the normal modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. If PARAM,W4 is not specified, GEi is ignored in transient analysis.
4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces.  $\sigma_i = F_i \cdot SA$  or  $\sigma_i = M_i \cdot ST$
5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements.  $\varepsilon_i = U_i \cdot EA$  or  $\varepsilon_i = \theta_i \cdot ET$
6. The "K", "B", "GE", "RCV", or "M" entries may be specified in any order.
7. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
8. Applicable fields refer to directions in the element's coordinate system.

9. For upward compatibility, if ONLY GE1 is specified on a PBUSH entry and GEi, i = 2 – 6 are all blank on the PBUSH entry, then the single structural damping GE1 is applied to all defined Ki for the PBUSH entry. In this case, GEi, i = 2 - 6, blank is not equivalent to 0.0 but a special flag.

For any PBUSH entry that has any GEi, i = 2 – 6 numerically specified including a 0.0, the fields GEi are all treated separately and any GEi, i = 2 - 6 field with no entry specified is defaulted to 0.0. In this case, for any GEi, i = 2 – 6, blank is equivalent to 0.0 and the results are radically different from the no GEi, i = 2- 6 specified as described above.

Thus for the entry:

| 1     | 2       | 3  | 4    | 5     | 6    | 7     | 8     | 9     | 10 |
|-------|---------|----|------|-------|------|-------|-------|-------|----|
| PBUSH | 3303000 | K  | 653. | 4000. | 460. | 10.+3 | 10.+3 | 10.+3 |    |
|       |         | GE | .05  |       |      |       |       |       |    |

The GE=.05 will be replicated to fields 5 through 9.

While for the entry:

| 1     | 2       | 3  | 4    | 5     | 6    | 7     | 8     | 9     | 10 |
|-------|---------|----|------|-------|------|-------|-------|-------|----|
| PBUSH | 3303000 | K  | 653. | 4000. | 460. | 10.+3 | 10.+3 | 10.+3 |    |
|       |         | GE | .05  | 0.0   |      |       |       |       |    |

The element x-direction will have a GE=.05 while the other 5 directions will have an associated GE=0.0

The entry:

| 1     | 2       | 3  | 4    | 5     | 6    | 7     | 8     | 9     | 10 |
|-------|---------|----|------|-------|------|-------|-------|-------|----|
| PBUSH | 3303000 | K  | 653. | 4000. | 460. | 10.+3 | 10.+3 | 10.+3 |    |
|       |         | GE | .05  |       | .02  |       |       |       |    |

is equivalent to the full entry:

| 1     | 2       | 3  | 4    | 5     | 6    | 7     | 8     | 9     | 10 |
|-------|---------|----|------|-------|------|-------|-------|-------|----|
| PBUSH | 3303000 | K  | 653. | 4000. | 460. | 10.+3 | 10.+3 | 10.+3 |    |
|       |         | GE | .05  | .0    | .02  | .0    | .0    | .0    |    |

IFPSTAR introduced an incompatibility with the above rules in Nastran versions 2014 to 2017. For Nastran version 2018 IFPSTAR, the above original rules have been restored. However, the previous incompatible IFPSTAR results can be recovered with bulk data entry MDLPRM, GEV1417, 1 or NASTRAN, SYSTEM(754)=1 in which case the last example becomes equivalent to the full entry:

| 1     | 2       | 3  | 4    | 5     | 6    | 7     | 8     | 9     | 10 |
|-------|---------|----|------|-------|------|-------|-------|-------|----|
| PBUSH | 3303000 | K  | 653. | 4000. | 460. | 10.+3 | 10.+3 | 10.+3 |    |
|       |         | GE | .05  | .05   | .02  | .05   | .05   | .05   |    |

10. For SOL 600, it is not necessary to enter PBUSH if a PBUSHT with the same ID is used in the model. Omitting the PBUSH entry if a PBUSHT with the same ID is entered.

11. For SOL 600, the defaults for SA, ST, EA and ET are 1.0E-10.
12. The mass properties are assumed located at the point defined by S (or S1, S2, S3 if OCID>-1) on the CBUSH entry. They are assumed to be in the element coordinate system. Define  $\alpha = S$  for the case OCID= -1 (default). See CBUSH entry. For the case OCID> -1, define  $r_A, r_B$  to be the vector from grid GA to (S1,S2,S3) and from grid GB to (S1,S2,S3) respectively, then

$$\alpha = \frac{|r_A|}{|r_A| + |r_B|}$$

$$|r_A| = \sqrt{(GA1 - S1)^2 + (GA2 - S2)^2 + (GA3 - S3)^2}$$

$$|r_B| = \sqrt{(GB1 - S1)^2 + (GB2 - S2)^2 + (GB3 - S3)^2}$$

Then the diagonal contribution to GA for the mass matrix in the element coordinate system is:

$$M11A = M22A = M33A = (1 - \alpha) * M$$

The diagonal contribution to GB for the mass matrix in the element coordinate system is:

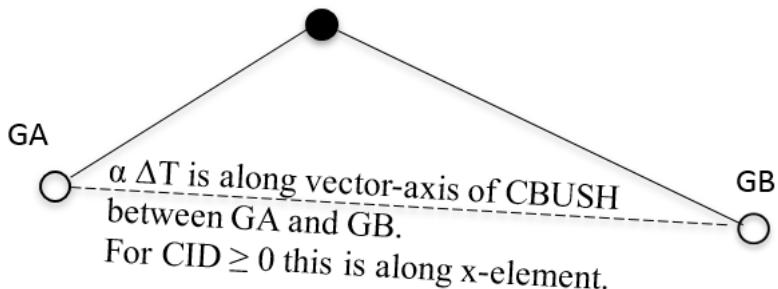
$$M11B = M22B = M33B = (\alpha) * M$$

13. PBUSH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH entries.
14. The CBUSH is a self-equilibrating element which means that for a CBUSH of finite length, its translational stiffness terms couple with rotational stiffness for rotational degrees of freedom. However, this coupling is dependent on the orientation of the CBUSH element. Thus there may be times when no explicit rotational stiffness is specified along a direction for which translational stiffness was specified that there is also no coupled rotational stiffness for that direction.

In modal analysis, a CBUSH of finite length having rotational degrees of freedom with no rotational stiffness (coupled or explicit) and no associated mass on the grids may cause massless mechanisms. A massless mechanism means the natural frequency for this rotational mode approaches the limit of zero rotational stiffness divided by zero mass, which is an undefined quantity. If the elements lie along a global coordinate axis, the mass term is identically zero, which leads to very large negative or positive eigenvalues and is usually beyond any reasonable search region. If the elements are skewed from the global axes, the eigenvalues may be computed at any value (including negative) because of the indeterminacy caused by numerical truncation.

15. The thermal expansion for the CBUSH will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERTURE Case Control command for thermal loading rules.

### Bush Location



**PBUSH1D**

## Rod Type Spring-and-Damper Property

Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).

**Format:**

| 1       | 2        | 3    | 4     | 5     | 6      | 7     | 8     | 9     | 10 |
|---------|----------|------|-------|-------|--------|-------|-------|-------|----|
| PBUSH1D | PID      | K    | C     | M     |        | SA    | SE    |       |    |
|         | "SHOCKA" | TYPE | CVT   | CVC   | EXPVT  | EXPVC | IDTS  |       |    |
|         |          |      | IDECS | IDECS | IDESTD | IDECS |       |       |    |
|         | "SPRING" | TYPE | IDT   | IDC   | IDTDU  | IDCDU |       |       |    |
|         | "DAMPER" | TYPE | IDT   | IDC   | IDTDV  | IDCDV |       |       |    |
|         | "GENER"  | TYPE | IDT   | IDC   | IDTDU  | IDCDU | IDTDV | IDCDV |    |

**Example:**

|         |        |       |      |      |    |  |     |  |  |
|---------|--------|-------|------|------|----|--|-----|--|--|
| PBUSH1D | 35     | 3000. | 200. | 300. |    |  |     |  |  |
|         | SHOCKA | TABLE | 2.2  | 1.2  | 1. |  | 200 |  |  |

The continuation entries are optional. The four options, SHOCKA, SPRING, DAMPER, and GENER can appear in any order.

| Descriptor | Meaning   | Default               |
|------------|---|-----------------------|
| PID        | Property identification number. (Integer > 0).            | Required              |
| K          | Stiffness. (Real $\geq 0$ ).                              | See Remark 1.         |
| C          | Viscous damping. (Real $\geq 0$ ).                        | See Remarks 1. and 2. |
| M          | Total mass of the element. (Real $\geq 0$ ).              | Blank                 |
| SA         | Stress recovery coefficient [1/area]. (Real $\geq 0$ ).   | Blank                 |
| SE         | Strain recovery coefficient [1/length]. (Real $\geq 0$ ). | Blank                 |

| Descriptor | Meaning  | Default                            |
|------------|--|------------------------------------|
| SHOCKA     | Character string specifying that the next 10 fields are coefficients of the following force versus velocity/displacement relationship. (Character).  |                                    |
|            | $F(u, v) = C_v \cdot S(u) \cdot \text{sign}(v) \cdot  v ^{\text{EXPV}}$  |                                    |
|            | The force F, the displacement u, and the velocity v, are in the axial direction of the damper. The axis of the damper is defined by the two connecting grid points GA and GB on the CBUSH1D Bulk Data entry. The displacement u and the velocity v, are the relative displacement and the relative velocity with respect to the grid point GA. The scale factor S(u) must be defined with a table or with an equation. |                                    |
| TYPE       | Character string indicating the type of definition. (Character). For TYPE = EQUAT, the fields IDETS, IDECS, IDETSD, and IDECSD are identification numbers of DEQATN entries. For TYPE = TABLE the field IDTS is an identification number of a TABLEDi entry. If no character string is provided (blanks), TYPE = TABLE is set.   | TABLE                              |
| CVT        | Viscous damping coefficient $C_V$ for tension $v > 0$ , force per unit velocity. (Real).   | Required for SHOCKA                |
| CVC        | Viscous damping coefficient $C_V$ for compression $v > 0$ , force per unit velocity. (Real).   | CVT                                |
| EXPVT      | Exponent of velocity EXPV for tension $v > 0$ . (Real).  | 1.                                 |
| EXPVC      | Exponent of velocity EXPV for compression $v < 0$ . (Real).  | EXPVT                              |
| IDTS       | Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE. The TABLEDi entry defines the scale factor S, versus displacement u.   | Required for SHOCKA and TYPE=TABLE |
| IDEATS     | Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for tension $u > 0$ .   | Required for SHOCKA and TYPE=EQUAT |
| IDECS      | Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for compression $u < 0$ .   | IDEATS                             |
| IDETSD     | Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for tension $u > 0$ .  | Required for SHOCKA and TYPE=EQUAT |

| Descriptor | Meaning   | Default                                |
|------------|---|--|
| IDECSID    | Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for compression $u < 0$ .   | IDETSD                                 |
| SPRING     | Character string specifying that the next 5 fields define a nonlinear elastic spring element in terms of a force versus displacement relationship. (Character).   |  |
|            | $F(u) = F_T(u)$   |  |
|            | Tension is $u > 0$ and compression is $u < 0$ .   |  |
| DAMPER     | Character string specifying that the next 5 fields define a nonlinear viscous element in terms of a force versus velocity relationship. (Character).  |  |
|            | $F(v) = F_T(v)$   |  |
|            | Tension is $v > 0$ and compression is $v < 0$ .   |  |
| GENER      | Character string specifying that the next 7 fields define a general nonlinear elastic spring and viscous damper element in terms of a force versus displacement and velocity relationship. (Character). For this element, the relationship can only be defined with TYPE=EQUAT.             |  |
|            | $F(u, v) = F_T(u, v)$   |  |
|            | Tension is $u > 0$ and compression is $u < 0$ . For SPRING, DAMPER, and GENER, the remaining fields are   |  |
| TYPE       | Character string indicating the type of definition. (Character). For TYPE = EQUAT the following fields are identification numbers of DEQATN entries. For TYPE = TABLE the following field is an identification number of a TABLEDi entry. For "GENER", TYPE = blank or "EQUAT" is allowed.. | Required for SPRING or DAMPER          |
| IDT        | Identification number of a DEQATN entry for tension if TYPE = EQUAT. Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE.  | Required for SPRING, DAMPER, and GENER |
| IDC        | Identification number of a DEQATN entry for compression if TYPE = EQUAT. Is ignored for TYPE = TABLE.   | IDT                                    |

| Descriptor | Meaning  | Default                |
|------------|--|------------------------|
| IDTDU      | Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for tension $u > 0$ . For SPRING and GENER only.         | Required if TYPE=EQUAT |
| IDCDU      | Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for compression $u < 0$ . For SPRING and GENER only. | IDTDU                  |
| IDTDV      | Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for tension $v > 0$ . For DAMPER and GENER only.             | Required if TYPE=EQUAT |
| IDCDV      | Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for compression $v < 0$ . For DAMPER and GENER only.     | IDCDT                  |

### Remarks:

1. For a linear spring K a default value of 0.0 is allowed by not specifying the K entry. For a linear damper C a default value of 0.0 is allowed by not specifying the C entry. If any nonlinearity is specified (SHOCKA, SPRING, DAMPER, GENER)  $K > 0$  is required. If K is not provided in the presence of any nonlinearity, the element will behave as a linear element.
2. The damping C and mass M are ignored in static solution sequences.
3. The parameters defined on the continuation entries are used in nonlinear solution sequences only.
4. The linear parameters K and C are used in all solution sequences unless parameters on continuation entries are defined and a nonlinear solution sequence is used. Then, the parameters K and C are used for initial values in the first iteration of the first load step and the parameters from continuation entries overwrite the linear parameters thereafter. When SHOCKA, SPRING or GENER are specified, K is overwritten. When SHOCKA, DAMPER or GENER is specified, C is overwritten.
5. PBUSH1D may only be referenced by CBUSH1D elements in the residual structure which do not attach to omitted degrees-of-freedom.
6. The continuation entries SHOCKA, SPRING, DAMPER and GENER may be specified in any order. If more than one continuation entry is defined, then the forces of SHOCKA, SPRING, etc. are added. Multiple continuation entries of the same kind are not allowed, for example, multiple SPRING continuation entries.
7. For TYPE = TABLE, values on the TABLEDi entry are for tension and compression. If table values  $f(u)$  are provided only for positive values  $u > 0$ , then it is assumed that  $f(-u) = -f(u)$ .
8. For TYPE = EQUAT, the equations for tension and compression can be different. If the identification numbers for compression are left blank, it is assumed that the equation for tension is also valid for compression.

9. PBUSH1D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH1D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH1D entries.

10. The linear FORCE (STRESS) output is:

AXIAL-VELOCITY, AXIAL-STRESS

For nonlinear FORCE (STRESS) output (Continuation entry on PBUSH1D.) the following addition output where applicable is:

AXIAL-VELOCITY, AXIAL-STRESS, AXIAL-STRAIN, PLASTIC-STRAIN,  
STATUS

All seven header titles appear in both linear and nonlinear.

11. While computing Single Point Forces of Constraint (SPCFORCES), the constraint relationships exist in stiffness for linear spring. However, the nonlinear elements treat the constraints as loads on set. As a result, the SPCFORCES results are different between linear and nonlinear elements.

**PBUSH2D**

## 2-D Linear/Nonlinear Connection Properties

Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).

**Format:**

| 1       | 2       | 3     | 4      | 5     | 6      | 7       | 8       | 9 | 10 |
|---------|---------|-------|--------|-------|--------|---------|---------|---|----|
| PBUSH2D | PID     | K11   | K22    | B11   | B22    | M11     | M22     |   |    |
|         | SQUEEZE | BDIA  | BLEN   | BCLR  | SOLN   | VISCO   | PVAPCO  |   |    |
|         | NPORT   | PRES1 | THETA1 | PRES2 | THETA2 | OFFSET1 | OFFSET2 |   |    |

**Example:**

|         |         |       |       |       |      |      |       |  |  |
|---------|---------|-------|-------|-------|------|------|-------|--|--|
| PBUSH2D | 1000    | 50.0  | 150.0 | 0.02  | 0.02 |      |       |  |  |
|         | SQUEEZE | 1.0   | 2.0   | 0.05  | LONG | 2.1  | 300.0 |  |  |
|         | 2       | 100.0 | 30.0  | 120.0 | 90.0 | 0.01 | 0.0   |  |  |

OR (to include cross coupling terms)

|         |       |     |     |     |     |     |     |  |  |
|---------|-------|-----|-----|-----|-----|-----|-----|--|--|
| PBUSH2D | PID   | K11 | K22 | B11 | B22 | M11 | M22 |  |  |
|         | CROSS | K12 | K21 | B12 | B21 | M12 | M21 |  |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number (Integer > 0; Required).                     |
| K11        | Nominal stiffness in T1 rectangular direction (Real; Required).             |
| K22        | Nominal stiffness in T2 rectangular direction (Real; Required).             |
| B11        | Nominal damping in T1 rectangular direction (Real; Default = 0.0).          |
| B22        | Nominal damping in T2 rectangular direction (Real; Default = 0.0).          |
| M11        | Nominal acceleration-dependent force in T1 direction (Real; Default = 0.0). |
| M22        | Nominal acceleration-dependent force in T2 direction (Real; Default = 0.0). |
| 'SQUEEZE'  | Indicates that squeeze-film damper will be specified (Character; Required). |
| BDIA       | Inner journal diameter. (Real > 0.0; Required)                              |
| BLEN       | Damper length. (Real > 0.0; Required).                                      |
| BCLR       | Damper radial clearance (Real > 0.0; Required).                             |
| SOLN       | Solution option: LONG or SHORT bearing (Character; Default = LONG).         |
| VISCO      | Lubricant viscosity (Real > 0.0; Required).                                 |
| PVAPCO     | Lubricant vapor pressure (Real; Required).                                  |
| NPORT      | Number of lubrication ports: 1 or 2 (Integer; no Default).                  |
| PRES1      | Boundary pressure for port 1 (Real $\geq$ 0.0; Required if NPORT= 1 or 2).  |

| Descriptor | Meaning  |
|------------|--|
| THETA1     | Angular position for port 1 ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT= 1 or 2). See Remark 2. |
| PRES2      | Boundary pressure for port 2 ( $\text{Real} \geq 0.0$ ; Required if NPORT= 2).                           |
| THETA2     | Angular position for port 2 ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT= 2). See Remark 2.      |
| OFFSET1    | Offset in the SFD direction 1, see Remark 3. (Real; Default = 0.0).                                      |
| OFFSET2    | Offset in the SFD direction 2, see Remark 3. (Real; Default = 0.0)                                       |
| CROSS      | Indicates that cross coupling terms will be provided.  |
| K12, K21   | Cross coupling stiffness terms.  |
| B12, B21   | Cross coupling damping terms.  |
| M12, M21   | Cross coupling mass terms.   |

**Remarks:**

1. Either SQUEEZE or CROSS option can be used. Both options cannot be specified.
2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the XZ plane.
3. Offsets are measured from GB relative to GA. For example, if direction 2 is in the vertical direction and a gravity load is placed on GA, OFFSET2 will be a positive value (GB ‘moves’ toward GA in the positive direction 2).
4. PBUSH2D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH2D entries.
5. PBUSH2D may be used in conjunction with THPAD obtained from ROMAC for rotordynamic applications. Also see [ELEMUDS](#) and [THPAD](#) bulk data entries.
6. The template used for specifying matrices using cross entries is:

$$[I]_{2X2} = \begin{bmatrix} I_{11} & I_{12} \\ I_{21} & I_{22} \end{bmatrix}$$

where I=K,B or M

**PBUSHT**

## Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

**Format:**

| 1      | 2   | 3    | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|-----|------|--------|--------|--------|--------|--------|--------|----|
| PBUSHT | PID | “K”  | TKID1  | TKID2  | TKID3  | TKID4  | TKID5  | TKID6  |    |
|        |     | “B”  | TBID1  | TBID2  | TBID3  | TBID4  | TBID5  | TBID6  |    |
|        |     | “GE” | TGEID1 | TGEID2 | TGEID3 | TGEID4 | TGEID5 | TGEID6 |    |
|        |     | “KN” | TKNID1 | TKIND2 | TKNID3 | TKIND4 | TKIND5 | TKIND6 |    |
|        |     |      | FDC    | FUSE   | DIR    | OPTION | LOWER  | UPPER  |    |
|        |     |      | FSRS   | LRGR   |        |        |        |        |    |

**Example:**

|        |    |   |    |  |  |  |  |  |  |
|--------|----|---|----|--|--|--|--|--|--|
| PBUSHT | 35 | K | 72 |  |  |  |  |  |  |
|        |    | B | 18 |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)  |
| “K”        | Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)  |
| TKIDI      | Identification number of a TABLEDI entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer $\geq 0$ ; Default = 0)   |
| “B”        | Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)   |
| TBIDI      | Identification number of a TABLEDI entry that defines the force per unit velocity damping vs. frequency relationship in directions 1 through 6. (Integer $\geq 0$ ; Default = 0)                                   |
| “GE”       | Flag indicating that the next field is a structural damping frequency table identification number. (Character)   |
| TGEIDI     | Identification number of a TABLEDI entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer $\geq 0$ ; TGEID1 default = 0, for TGEID2-6 see defaults described in Remark 6.) |
| “KN”       | Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)   |
| TKNIDI     | Identification number of a TABLEDI entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer $\geq 0$ ; Default = 0)  |

| Descriptor    | Meaning   |
|---------------|---|
| FDC           | Force deflection curve rule. Specifies a dependence between displacement components. See Remark 16. for use with the FUSE option. (Character or blank default blank)  |
| "NR" or blank | Implies there is no force deflection rule. T1-R6 directions are all independent of each other.  |
| "TRXY"        | A radial dependence exists between $u_x$ and $u_y$ element displacements. Either K1 or K2 may be specified on the PBUSH entry. If both are specified then K1 will be used. On the PBUSHT "KN" entry either TKNID1 or TKNID2 may be specified. If both are specified then TKNID1 is used. The expression                       |
|               | $u_r = \sqrt{(u_x)^2 + (u_y)^2}$  |
|               | is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_x$ and $f_y$ in the springs "x" and "y" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current $u_r$ . |
| "TRXZ"        | A radial dependence exists between $u_x$ and $u_z$ element displacements. Either K1 or K3 may be specified on the PBUSH entry. If both are specified then K1 is used. On the PBUSHT "KN" entry either TKNID1 or TKNID3 may be specified. If both are specified then TKNID1 is used. The expression                            |
|               | $u_r = \sqrt{(u_x)^2 + (u_z)^2}$  |
|               | is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_x$ and $f_z$ in the springs "x" and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current $u_r$ . |
| "TRYZ"        | A radial dependence exists between $u_y$ and $u_z$ element displacements. Either K2 or K3 may be specified on the PBUSH entry. If both are specified then K2 is used. On the PBUSHT "KN" entry either TKNID2 or TKNID3 may be specified. If both are specified then TKNID2 is used. The expression                            |
|               | $u_r = \sqrt{(u_y)^2 + (u_z)^2}$  |
|               | is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_y$ and $f_z$ in the springs "y" and "z" are used for the fuse calculation. New K2 is returned from slope of load-deflection curve at slope at current $u_r$ . |

| Descriptor | Meaning   |
|------------|---|
|            | "TS"  |
|            | A spherical dependence exists between $u_x$ , $u_y$ , and $u_z$ element displacements. Either K1, K2 or K3 may be specified on the PBUSH entry. If all three are specified K1 is used. IF only K2 and K3 are specified then K2 is used. On the PBUSH "KN" entry TKNID1, or TKNID2 or TKNID3 may be specified. If all three are specified then TKNID1 is used. If only TKNID2 and TKNID3 are specified then TKNID2 is used. The expression<br>$u_s = \sqrt{(u_x)^2 + (u_y)^2 + (u_z)^2}$ is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_s$ . Under the OPTION=RULTLD, the true forces $f_x$ , $f_y$ and $f_z$ in the springs "x", "y", and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current $u_s$ |
| FUSE       | 0<br>The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)<br>1<br>The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for postprocessing.<br>2<br>The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from postprocessing.   |
| DIR        | The fuse direction component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks; Default = 0)<br>0<br>Any direction is allowed to fuse. Same if user entered 123456.<br>1 through 6 may be placed in the field with no embedded blanks. See Remark 13. for use with FDC.  |
| OPTION     | Specifies a failure mode. (Character - Default="RELDIS")<br>"ULTLD" The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship.<br>"RELDIS" The specified max relative $\pm$ displacement will be used to define failure.<br>"RULTLD" The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.   |
| LOWER      | Lower failure bound. (Real; Default=0.0)<br>If OPTION = "ULTLD" then LOWER specifies a lower failure load<br>If OPTION = "RELDIS" then LOWER specifies a minimum relative displacement before failure.  |
| UPPER      | Upper failure bound. (Real; Default=0.0)  |

| Descriptor | Meaning  |
|------------|--|
|            | If OPTION = "ULTLTD" then UPPER specifies an upper failure load.   |
|            | If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.  |
| FSRS       | Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real $\geq 0.0$ ; Default = 1.-5) |
| LRGR       | Specifies if large rotation is to occur at end A. (Integer $\geq 0$ ; Default = 0). See Remark 14.   |

**Remarks:**

1. The "K", "B", and "GE" fields are associated with same entries on the PBUSH entry.
2. PBUSHT may only be referenced by CBUSH elements in the residual structure which do not attach to any omitted degrees-of-freedom of ASET and/or dependent DOFs of MPC/Rigid elements. Otherwise, the solution is an approximation based on nominal properties.
3. For nonlinear analysis the nominal values are used at the beginning of the analysis to compute initial loading of the CBUSH element and meaningful values should be supplied for stiffness. As the nonlinear analysis proceeds, the user supplied load deflection curves are used to compute new stiffness values. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
4. The "K", "B", "GE" or "KN" fields may be specified in any order.
5. The PBUSHT is used in SOL 108 and SOL 111 when any "K", "B", or "GE" is specified. It is used in SOL 106, SOL 129 or SOL 400 when "KN" is specified. It is ignored in all other solution sequences.
6. For upward compatibility, if ONLY TGEID1 is specified on a PBUSHT entry and TGEIDI<sub>i</sub>, i = 2 – 6 are all blank on the PBUSHT entry, then the single structural damping table is applied to all defined GEi for the PBUSH entry. In this case, TGEIDI<sub>i</sub>, i = 2 – 6, blank is not equivalent to 0 but a special flag.

A PBUSHT entry that has any TGEIDI<sub>i</sub>, i = 2 – 6 numerically specified including a 0, all the fields TGEIDI<sub>i</sub> are then treated separately and any TGEIDI<sub>i</sub>, i=2 - 6 field with no entry specified is defaulted to a 0 value and the results are radically different from the no TGEIDI<sub>i</sub>, i=2 – 6 specified as described above.

Thus for the entry:

| 1      | 2       | 3  | 4        | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---------|----|----------|---|---|---|---|---|----|
| PBUSHT | 3303000 | K  | 33030001 |   |   |   |   |   |    |
|        |         | GE | 33030002 |   |   |   |   |   |    |

The GE table ID 3303002 will be replicated to fields 5 through 9. While:

| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|----|
|   |   |   |   |   |   |   |   |   |    |

|        |         |    |          |   |  |  |  |  |
|--------|---------|----|----------|---|--|--|--|--|
| PBUSHT | 3303000 | K  | 33030001 |   |  |  |  |  |
|        |         | GE | 33030002 | 0 |  |  |  |  |

The 0 in FIELD 5 of the above PBUSHT above negates table 3303002 being replicated, hence, fields 5 through 9 will not have any associated table lookup.

7. For nonlinear analysis, only the “KN” line and optionally its two continuation lines are used.
8. For frequency responses, only the “K”, “B” and/or “GE” fields are used.
9. The continuations entries to “KN” are optional.
10. When the FDC field specifies one of the optional radial or spherical rules, stiffness entries not involved in the rule may still have their own independent force-deflection curves.
11. If FUSE=0, the remaining entries are ignored. If FUSE > 0, the “OPTION”, “UPPER”, and “LOWER” fields must be specified.
12. UPPER > LOWER required.
13. The following relationships exists between FDC and DIR

|              |   |                                       |
|--------------|---|---------------------------------------|
| FDC = “TRXY” | 1 or 2 or 12 in the DIR field applies to the radial rule  | a 3 in the DIR field will be ignored. |
| FDC = “TRXZ” | 1 or 3 or 13 in the DIR field applies to the radial rule  | a 2 in the DIR field will be ignored. |
| FDC = “TRYZ” | 2 or 3 or 23 in the DIR field applies to the radial rule  | a 1 in the DIR field will be ignored. |
| FDC = “TS”   | Any unique combination of the integers 1 through 3 with no embedded blanks in the DIR field applies to the spherical rule |                                       |

14. LRGR = 0 (Default) the element coordinate system is rotated with the rotation of grid A for both the CID and the V vector. LRGR = 1 will suppress large rotation at end A. The initial CID and the V vector will remain unchanged. LRGR = 2 a mid-increment method is used to rotate the element system for the V vector. LRGR = 2 is deactivated if CID is not blank. When CID is blank, LRGR = 2 is recommended for SOL400 when large rotations are present as the mid-increment method “averages” the rotations at both ends of the CBUSH and allows rotation when end A might have a rotational constraint.
15. When the fuse criteria is reached in one of the stiffness components, the entire element is assumed to fail. The component stiffnesses are then computed to be  $K = K \cdot FSRS$  at initial failure.
16. The figures below depict, using FDC=TRYZ as a typical example, the OPTION="ULTLD" and OPTION="RULTLD". Under the OPTION="ULTLD" the resultant Fr load when FUSED will be  $F_r = \sqrt{F_y^2 + F_z^2} \approx \text{LIMIT}$ . Under the OPTION="RULTLD" the resultant Fr load when FUSED will in general be  $F_r = \sqrt{F_y^2 + F_z^2} > \text{LIMIT}$  with one of the  $F_y$  or  $F_z \approx \text{LIMIT}$ . For FDC=TRYZ and

DIR=123, the T1 direction is not included in the force direction curve rule, thus possible fusing in the T1 direction is always based on the actual T1 spring displacement and  $F_x$  if active will be  $F_x \approx \text{LIMIT}$ . For FDC=NR fusing is always calculated based on each individual spring displacement and the actual individual spring loading.

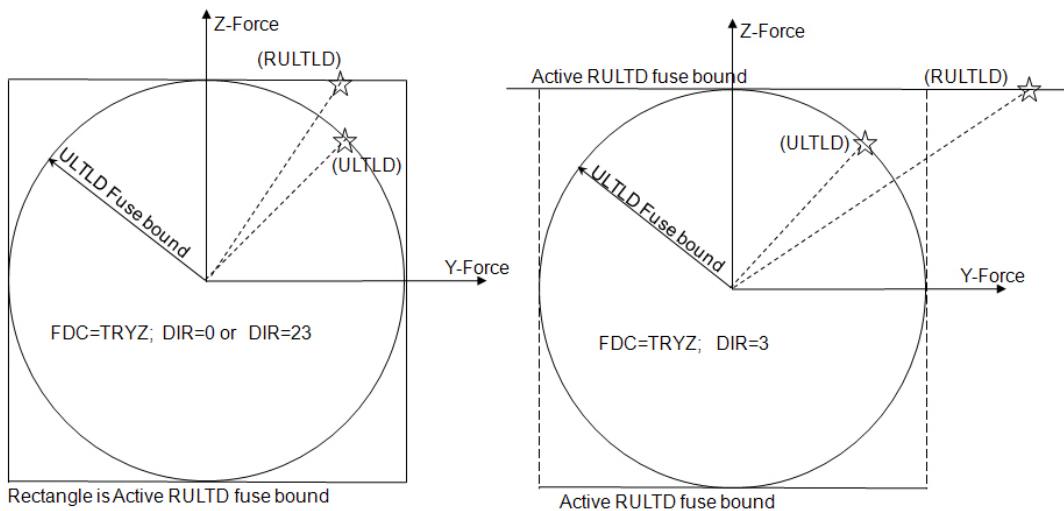


Figure 9-132 FUSE Models

**PCOHE****Interface Cohesive Zone Modeling Element Properties**

Defines the properties of a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400.

**Format:**

| 1     | 2   | 3   | 4   | 5 | 6      | 7      | 8 | 9 | 10 |
|-------|-----|-----|-----|---|--------|--------|---|---|----|
| PCOHE | PID | MID | INT | T | OUTPUT | SECANT |   |   |    |

**Example:**

|       |     |     |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|
| PCOHE | 700 | 701 |  |  |  |  |  |  |  |
|-------|-----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification. (Integer > 0)  |
| MID        | Identification number of a MCOHE entry. (Integer > 0)   |
| INT        | Integration scheme. 0 or blank Gauss integration, 1 for Newton-Coates/Lobatto integration. See Remark 3. (Integer 0 or 1; Default 0)  |
| T          | Thickness. See Remark 2. (Real > 0; Default = 1.0)  |
| OUTPUT     | Location selection for stress/strain output. If OUTPUT=GRID or blank, output is at the corner grid points. If OUTPUT=GAUSS output at the Gauss points. If INT=1, OUTPUT will be set to GRID. (Character or Blank; Default Grid) |
| SECANT     | Tangent matrix scheme. 0 or blank a secant-type matrix is used to set up the element stiffness matrix, 1 a tangent-type matrix is used to set up the element stiffness matrix. (Integer 0 or 1; Default 0)                      |

**Remarks:**

1. PCOHE can be referenced by CIFQUAD, CIFHEX, CIFPENT, and CIFQDX entries.
2. The thickness T applies only to CIFQUAD elements.
3. For initially very stiff interface elements, the Newton-Coates/Lobatto integration scheme may be preferred.
4. PCOHE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOHE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PCOHE entries

**PCOMP**

## Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

**Format:**

| 1     | 2    | 3  | 4      | 5     | 6      | 7    | 8      | 9     | 10 |
|-------|------|----|--------|-------|--------|------|--------|-------|----|
| PCOMP | PID  | Z0 | NSM    | SB    | FT     | TREF | GE     | LAM   |    |
|       | MID1 | T1 | THETA1 | SOUT1 | MID2   | T2   | THETA2 | SOUT2 |    |
|       | MID3 | T3 | THETA3 | SOUT3 | -etc.- |      |        |       |    |

**Example of multiple plies per line format:**

|       |     |        |      |         |      |  |     |  |  |
|-------|-----|--------|------|---------|------|--|-----|--|--|
| PCOMP | 181 | -0.224 | 7.45 | 10000.0 | HOFF |  |     |  |  |
|       | 171 | .056   | 0.   | YES     |      |  | 45. |  |  |
|       |     |        | -45. |         |      |  | 90. |  |  |

**Example of single ply per line format:**

|       |     |        |      |        |      |  |  |  |  |
|-------|-----|--------|------|--------|------|--|--|--|--|
| PCOMP | 181 | -0.224 | 7.45 | 10000. | HOFF |  |  |  |  |
|       | 171 | .056   | 0.   | YES    |      |  |  |  |  |
|       | 171 | .056   | 45.  | YES    |      |  |  |  |  |
|       | 171 | .056   | -45. | YES    |      |  |  |  |  |
|       | 171 | .056   | 90.  | YES    |      |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (0 < Integer < 10000000)  |
| Z0         | Distance from the reference plane to the bottom surface. See Remarks 10. and 11. (Real; Default = -0.5 times the element thickness.)  |
| NSM        | Nonstructural mass per unit area. (Real)  |
| SB         | Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)  |
| FT         | Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 7.<br>“HILL” for the Hill theory.<br>“HOFF” for the Hoffman theory.<br>“TSAI” for the Tsai-Wu theory.<br>“STRN” for the Maximum Strain theory.<br>“HFFAIL” for the Hashin failure criterion<br>“HTAPE” for the Hashin tape criterion<br>“HFABR” for the Hashin fabric criterion |
| TREF       | Reference temperature. See Remark 3. (Real; Default = 0.0)  |

| Descriptor        | Meaning  |
|-------------------|--|
| GE                | Damping coefficient. See Remarks 4. and 12. (Real; Default = 0.0)  |
| LAM               | Laminate Options. (Character or blank, Default = blank). See Remarks 13. and 14. <ul style="list-style-type: none"> <li>“Blank” All plies must be specified and all stiffness terms are developed.</li> <li>“SYM” Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1) should be half the actual thickness.</li> <li>“MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.</li> <li>“BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.</li> <li>“SMEAR” All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set as blanks).</li> <li>“SMCORE” All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.</li> </ul> |
| MID <sub>i</sub>  | Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, MAT8, or MATDIGI Bulk Data entries. See Remarks 1. 15. and 18. (Integer > 0 or blank, except MID1 must be specified.)   |
| T <sub>i</sub>    | Thicknesses of the various plies. See Remarks 1. (Real or blank, except T1 must be specified.)   |
| THETAI            | Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)   |
| SOUT <sub>i</sub> | Stress or strain output request. See Remarks 5. and 6. (Character: “YES” or “NO”; Default = “NO”)  |

**Remarks:**

1. The default for MID<sub>2</sub>, ..., MID<sub>n</sub> is the last defined MID<sub>i</sub>. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
2. At least one of the four values (MID<sub>i</sub>, Ti, THETAI, SOUT<sub>i</sub>) must be present for a ply to exist. The minimum number of plies is one.
3. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMP entry is used for all plies of the element. If not specified, it defaults to “0.0.”

If the PCOMP references temperature dependent material properties, then the TREF given on the PCOMP will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMP entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL, TREF is not applicable.

4. GE given on the PCOMP entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMP entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GEij values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMP entry and use the given GEij values.
5. Stress and strain output for individual plies are available in all superelement static, normal modes, buckling, frequency response (direct and modal), transient response (direct and modal) and nonlinear static analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XYPILOT command.
6. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark [10](#).
7. In order to get failure index output the following must be present:
  - a. STRESS or STRAIN Case Control commands,
  - b. SB, FT, and SOUTi on the PCOMP Bulk Data entry,
  - c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
  - d. -1 - failure in the fiber direction.  
-2 - failure in the matrix direction  
-12 - failure in the inplane shear
  - e. For Hashin Failure criterion, HF<sub>i</sub> ( $i=1,2,3,4,10,11$ ) must be present in all referenced MAT8 bulk entry.
  - f. For Hashin Tape criterion, HT<sub>i</sub> ( $i=1,2,3,4,5,6,10,11,12$ ) must be present in all referenced MAT8 bulk entry.
  - g. For Hashin Fabric criterion, HF<sub>Bi</sub> ( $i=1,2,3,4,5,6,10,11,12$ ) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
  - 1 - failure in the tensile first fiber mode.
  - 1 - failure in the compressive first fiber mode.
  - 2 - failure in the tensile second fiber mode.
  - 2 - failure in the compressive second fiber mode.
  - 3 - failure in the tensile matrix mode.
  - 3 - failure in the compressive matrix mode.
8. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
9. The failure index for the bonding material is calculated as Failure Index =  $(\tau_{1z}, \tau_{2z})/SB$ .
10. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.
11. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
12. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
13. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide special purpose stiffness calculations. SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared. SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.
14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
15. Temperature-dependent ply properties only available in SOL 106 and SOL 400. See PARAM,COMPMATT for details.
16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.
17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMP entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.

18. PCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.
19. For PCOMP, the non-default Zo option should not be used in conjunction with MEM, BEND, SMEAR or SMCORE as wrong results may occur. These four options provide special purpose stiffness calculations wherein stacking sequence effects are ignored (membrane-bending coupling terms are set to zero) at the preliminary design stage level. A better choice for offsets is to use the ZOFF option on the connectivity entries (CQUAD4, CTRIA3 etc).  
A User Fatal Message is issued if the user specifies a non-default value of Zo for any of the above smearing options. The fatal message may be changed to a User Warning Message by specifying a positive value for system cell 668 (e.g. nastran system(668)=1) to allow the program to continue.
20. In layer composite output, the traditional element model outputs stress or strain separately by using STRESS or STRAN Case Control Command, respectively.  
The advanced element model does not recognize STRAIN Case Control Command. As long as user gives STRESS Case Control Command, both stress and strain are outputted together (similar to NLSTRESS).

**PCOMPA**

## Additional Data for Layered Composite Element Property - SOL 700

Defines additional properties of a multi-ply laminate composite material. Used in SOL 700 only.

**Format:**

| 1      | 2       | 3    | 4      | 5   | 6      | 7    | 8       | 9   | 10 |
|--------|---------|------|--------|-----|--------|------|---------|-----|----|
| PCOMPA | PID     | FORM | SHFACT | REF | STRDEF | DT1D | STRNOUT | CLT |    |
| +      | SPINCOR |      |        |     |        |      |         |     |    |

**Example:**

|        |     |     |  |  |  |  |  |   |  |
|--------|-----|-----|--|--|--|--|--|---|--|
| PCOMPA | 1   | BLT |  |  |  |  |  | 1 |  |
| +      | YES |     |  |  |  |  |  |   |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Unique property number referring to a PCOMP property number. (Integer > 0; Required)  |
| FORM       | Element formulation. (Character; See Remark 1.)   |
| SHFACT     | Shear correction factor, see Remark 4. (Real $\geq 0.0$ ; default=0.83333)  |
| REF        | Reference surface. (Character; default=MID) <ul style="list-style-type: none"> <li>TOP Reference surface is the top of the surface.</li> <li>MID Reference surface is the central surface.</li> <li>BOT Reference surface is the bottom surface.</li> </ul> |
| STRDEF     | Definition in stress-strain output. (Character; default=FIBER) <ul style="list-style-type: none"> <li>FIBER Stresses defined in the fiber and matrix directions.</li> <li>ELEM Stresses defined in the element coordinate system</li> </ul>                 |
| DT1D       | Time step skip for one-dimensional failure modes. See Remark 2. (Character; default=NO) <ul style="list-style-type: none"> <li>YES Skip one-dimensional failure modes.</li> <li>NO Normal time-step calculation.</li> </ul>                                 |
| STRNOUT    | Strain output option. See Remarks 3. and 4. (Character; default=YES) <ul style="list-style-type: none"> <li>YES Total strain is calculated.</li> <li>NO No strain is stored in memory.</li> </ul>   |
| ICLT       | Option to use Classical Lamination Theory. See Remark 4. (Integer > 0; default=0) <ul style="list-style-type: none"> <li>1 Use the Classical Lamination Theory.</li> <li>0 Use the integration technique.</li> </ul>  |
| SPINCOR    | Spin correction. See Remark 4. (Character; default=NO)  |

| Descriptor | Meaning            |
|------------|--------------------|
| NO         | No SPINCOR applied |
| YES        | SPINCOR applied    |

**Remarks:**

1. For CQUAD4 elements, the default formulation is Key-Hoff. For CTRIA3 elements, the default formulation is C0-TRIA.
2. If the failure mode is such that fiber and shear strength or matrix and shear strength are lost in all layers, the element is not included in the time-step calculation. If the element fails completely, the element is omitted from the time-step calculations, irrespective of the value entered in this field.
3. If the STRNOUT field is NO, the strain cannot be output.
4. If ICLT is set to 1, the analysis is performed with classical lamination theory. In this case, it is not possible to request the total strain output. The (transverse) shear correction factor input is ignored since it is calculated inside Nastran. There is no update of the cross-sectional properties due to failure. The failure flag only indicates that the failure condition is satisfied. Additional output for element variables is available, namely the stress resultants (NXN, NYN, NXZ, MXN, MYN, MZN, QXZ, and QZZ). Also the ABD-Q matrices of each element can be requested for output. These data are only stored in the first layer. The variable names are AijM, BijM, DijM, and QsijM for the components of the A-, B-, D- and Q-matrices, respectively. For example, to request the A11 of the A-matrix, the variable name is A11M01.
5. The options for SPINCOR are:

NO      No SPINCOR correction is applied.

YES     A SPINCOR correction is applied.

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

The SPINCOR option is required for fabric models and is turned on by default to accurately keep track of the fiber directions.

**PCOMPF****Integration Procedure Used in Conjunction with PCOMP or PCOMPG**

Defines the integration procedure for through the thickness integration of composite shells in SOL 600 and SOL 400 only.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6  | 7 | 8 | 9 | 10 |
|--------|-----|------|------|------|----|---|---|---|----|
| PCOMPF | INT | PID1 | THRU | PID2 | BY | N |   |   |    |

**Alternate Formats (SOL 400):**

| 1      | 2    | 3     | 4    | 5    | 6    | 7     | 8     | 9     | 10 |
|--------|------|-------|------|------|------|-------|-------|-------|----|
| PCOMPF | INT  | PID1  | PID2 | PID3 | THRU | PID4  | PID5  | THRU  |    |
|        | PID6 | PID7  | TO   | PID8 | PID9 | PID10 | PID11 | PID12 |    |
|        | THRU | PID13 | BY   | N    |      |       |       |       |    |

| 1      | 2   | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|---|---|---|---|---|---|----|
| PCOMPF | INT | ALL |   |   |   |   |   |   |    |

**Examples:**

|        |   |     |      |     |    |    |  |  |  |
|--------|---|-----|------|-----|----|----|--|--|--|
| PCOMPF | 2 | 100 | THRU | 200 | BY | 10 |  |  |  |
|--------|---|-----|------|-----|----|----|--|--|--|

**Example of Application to Single PID (SOL 400):**

|        |     |      |    |    |    |  |    |  |  |
|--------|-----|------|----|----|----|--|----|--|--|
| PCOMPF | 1   |      | 23 | TO | 25 |  | 33 |  |  |
|        | 44  | THRU | 54 | BY | 2  |  |    |  |  |
|        | 100 |      |    |    |    |  |    |  |  |

| Descriptor | Meaning  |
|------------|--|
| INT        | 1      Conventional through the thickness integration of each layer, allows all available material behavior through the thickness. (Default)<br>2      Linear elastic material, fast-integrated through the thickness - thermal strains and temperature dependent material properties are not allowed.<br>3      Linear elastic material, fast integrated through the thickness. |
| PIDI       | Property identification number. (0 < Integer < 10000000) corresponds to a matching PCOMP or PCOMPG entry.  |
| N          | Property identification number increment. See Remark 7.. (Integer or blank)  |

**Remarks:**

1. If nonlinear behavior is set on a MATS1 (or other option), but INT is > 1, then the nonlinear material behavior is ignored.
2. In SOL 600 only, if temperature dependent behavior is specified on a MATT1 or similar option and INT=2, the material values specified on the MATT1, MATT2, MATT8 option are ignored (the values on MAT1, MAT2 and MAT8 are used). In SOL 400, temperature loading or temperature dependent behavoir should not be used with INT=2, a user fatal will be issued in this case. INT=3 can be used instead.
3. If more than one PCOMP exists with different INT values, and there is an overlap in PID's, that is to say a given PID could have been assigned different values of INT, then a user fatal will be issued.
4. With the 'THRU" and "THRU", "BY" forms, blank fields are allowed for readability. Any combination of a list of ID's and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing ID's. That is the list of ID's in a THRU range need not be continuous.
5. In SOL 600 only, if all composite shells are to use the same INT value, it may be entered with PARAM,MFASTCMP instead of PCOMP.
6. In SOL 600 only, if a "THRU" or "THRU", "BY" range is entered, all items associated with the range must be on the same line (or for large field a line and the continuation entry of that line).
7. For automatic generation of property identification numbers, the default increment value is 1 if property identification numbers are increasing or -1 if property identification numbers are decreasing (i.e., the user need not specify BY and the increment value).

**PCOMPQF****Frequency Dependent Composite structural damping Property**

Defines the frequency dependent properties for a PCOMP/PCOMPG Bulk Data entry.

**Format:**

| 1       | 2   | 3 | 4 | 5 | 6 | 7 | 8  | 9 | 10 |
|---------|-----|---|---|---|---|---|----|---|----|
| PCOMPQF | PID |   |   |   |   |   | GE |   |    |

**Examples:**

|         |    |  |  |  |  |  |    |  |  |
|---------|----|--|--|--|--|--|----|--|--|
| PCOMPQF | 33 |  |  |  |  |  | 15 |  |  |
|---------|----|--|--|--|--|--|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number that matches the identification number on PCOMP/PCOMPG entry. (Integer > 0)  |
| GE         | Identification number of a TABLEDi entry that defines the non-dimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0) |

**Remarks:**

1. The Fields 8 of this entry corresponds to Field 8 of a PCOMP or PCOMG entry. The value in Field 8 of the PCOMP or PCOMPG entry is replaced by the table referenced in the corresponding field of this entry.
2. If the PCOMP/PCOMG MIDi fields point to MAT2 entries with the 2nd continuation fields containing GEij entries, then this entry is ignored for that MIDi entry and GE frequency dependency must be indicated by use of the MAT2F entry with same ID as the MIDi entry.
3. IF GE=0.0 on corresponding PCOMP or PCOMPG then GE table must be blank or 0.

**PCOMPG**

## Layered Composite Element Property (Alternate to PCOMP Entry)

Defines global (external) ply IDs and properties for a composite material laminate.

**Format:**

| 1      | 2       | 3    | 4   | 5      | 6     | 7     | 8  | 9   | 10 |
|--------|---------|------|-----|--------|-------|-------|----|-----|----|
| PCOMPG | PID     | Z0   | NSM | SB     | FT    | TREF  | GE | LAM |    |
|        | GPLYID1 | MID1 | T1  | THETA1 | SOUT1 | GEFLG |    |     |    |
|        | GPLYID2 | MID2 | T2  | THETA2 | SOUT2 |       |    |     |    |

**Example of single ply per line format:**

|        |      |        |      |        |      |  |  |  |  |
|--------|------|--------|------|--------|------|--|--|--|--|
| PCOMPG | 181  | -0.224 | 7.45 | 10000. | HOFF |  |  |  |  |
|        | 1001 | 171    | .056 | 0.     | YES  |  |  |  |  |
|        | 101  | 171    | .07  | 45.    | YES  |  |  |  |  |
|        | 2002 | 171    | .056 | -45.   | YES  |  |  |  |  |
|        | 102  | 171    | 0.55 | 90.    | YES  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (0 < Integer < 10000000)   |
| Z0         | Distance from the reference plane to the bottom surface. See Remark 11. (Real; Default = -0.5 times the element thickness.)  |
| NSM        | Nonstructural mass per unit area. (Real)   |
| SB         | Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)   |
| FT         | Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 8.<br>“HILL” for the Hill theory.<br>“HOFF” for the Hoffman theory.<br>“TSAI” for the Tsai-Wu theory.<br>“STRN” for the Maximum Strain theory.<br>“HFAIL” for the Hashin failure criterion<br>“HTAPE” for the Hashin tape criterion<br>“HFABR” for the Hashin fabric criterion |
| TREF       | Reference temperature. See Remark 4. (Real; Default = 0.0)   |
| GE         | Damping coefficient. See Remark 5. (Real; Default = 0.0)   |
| LAM        | Laminate Options. (Character or blank, Default = blank). See Remarks 14.<br>“Blank” All plies must be specified and all stiffness terms are developed.   |

| Descriptor          | Meaning  |
|---------------------|--|
| "MEM"               | All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.  |
| "BEND"              | All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.   |
| "SMEAR"             | All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero.  |
| "SMCORE"            | All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.                       |
| GPLYID <sub>i</sub> | User-defined Global (External) Ply ID. See Remark 1. (Integer > 0)   |
| MID <sub>i</sub>    | Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 2. and 15. (Integer > 0 or blank, except MID1 must be specified.)  |
| T <sub>i</sub>      | Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)  |
| THETAI <sub>i</sub> | Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0) |
| SOUT <sub>i</sub>   | Stress or strain output request. See Remarks 6. and 7. (Character: "YES" or "NO"; Default = "NO")  |
| GEFLG               | Ply structural damping flag may be only entered only once and if entered must be entered on the first ply. (Integer -2,or -1; Default=0) See Remark 19.  |

### Remarks:

1. The global ply identification number must be unique with respect to other plies in the entry. The plies are defined in stacking sequence starting with the bottom layer.
2. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
3. The global ply ID (GPLYID<sub>i</sub>) and at least one of the four values (MID<sub>i</sub>, Ti, THETAI<sub>i</sub>, SOUT<sub>i</sub>) must be present for a ply to exist. The minimum number of plies is one.
4. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMPG entry is used for all plies of the element. If not specified, it defaults to "0.0."

If the PCOMPG references temperature dependent material properties, then the TREF given on the PCOMPG will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMPG entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL, TREF is not applicable.

5. GE given on the PCOMPG entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMPG entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GEij values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMPG entry and use the given GEij values.
6. Stress and strain output for individual plies are available in all superelement static, frequency response (direct and modal), transient response (direct and modal) and normal modes analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XYPILOT command.
7. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark 11.
8. In order to get failure index output the following must be present:
  - a. STRESS or STRAIN Case Control commands,
  - b. SB, FT, and SOUTi on the PCOMPG Bulk Data entry,
  - c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
  - d. -1 - failure in the fiber direction  
-2 - failure in the matrix direction  
-12 - failure in the inplane shear.
  - e. For Hashin Failure criterion, HFi ( $i=1,2,3,4,10,11$ ) must be present in all referenced MAT8 bulk entry.
  - f. For Hashin Tape criterion, HTi ( $i=1,2,3,4,5,6,10,11,12$ ) must be present in all referenced MAT8 bulk entry.
  - g. For Hashin Fabric criterion, HFBi ( $i=1,2,3,4,5,6,10,11,12$ ) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
  - 1 - failure in the tensile first fiber mode.
  - 1 - failure in the compressive first fiber mode.
  - 2 - failure in the tensile second fiber mode.
  - 2 - failure in the compressive second fiber mode.
  - 3 - failure in the tensile matrix mode.
  - 3 - failure in the compressive matrix mode.
- 9. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
- 10. The failure index for the bonding material is calculated as Failure Index =  $(\tau_{1z}, \tau_{2z})/SB$ .
- 11. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane.
- 12. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
- 13. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
- 14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
- 15. Temperature-dependent ply properties only available in SOL 106 and SOL400. See PARAM,COMPMATT for details.
- 16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.
- 17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMPF entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.
- 18. PCOMPG is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPG property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.

19. Defines structural damping matrix data for composites. This entry if used MUST appear on the first ply entry ONLY and all subsequent plies follows depending if -1 or -2 described below

| GEFLG Value | Meaning  |
|-------------|--|
| 0           | The GE value specified in Field 8 of the PCOMPG parent entry is used for computation of all structural damping.  |
| -1          | The GEi value associated with the MID1 of this ply and the MIDI with MID(i > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MIDI for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry, the GE value specified in Field 8 of the PCOMPG parent entry will be used. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping.                         |
| -2          | The GEi value associated with the MID1 of this ply and the MIDI with MID(i > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MIDI for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry then that ply will not contribute to the computation of the K4 structural damping of the composite. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping. |

When GEFLG is used, the smeared GE value will appear in the second continuation entry field (6) of the four internally generated equivalent MAT2 entries

If GEFLG appears with extended MAT2 entries the GEFLG is ignored for that PCOMPG: See Remark of MAT2 entry.

**PCOMPLS****Layered Solid Composite Element Property**

Defines global (external) ply IDs and properties for a composite material laminate in SOL 600, SOL 400, all linear solution sequences between SOL 101 and SOL 112, and analysis only for SOL 200.

**Format:**

| 1       | 2     | 3      | 4     | 5      | 6      | 7 | 8 | 9 | 10 |
|---------|-------|--------|-------|--------|--------|---|---|---|----|
| PCOMPLS | PID   | DIRECT | CORDM | SB     | ANAL   |   |   |   |    |
|         | “C8”  | BEH8   | INT8  | BEH8H  | INT8H  |   |   |   |    |
|         | “C20” | BEH20  | INT20 | BEH20H | INT20H |   |   |   |    |
|         | ID1   | MID1   | T1    | THETA1 |        |   |   |   |    |
|         | ID2   | MID2   | T2    | THETA2 |        |   |   |   |    |

**Example:**

|         |      |     |    |      |  |  |  |  |  |
|---------|------|-----|----|------|--|--|--|--|--|
| PCOMPLS | 782  | 1   |    |      |  |  |  |  |  |
|         | 1001 | 171 | .3 | 12.3 |  |  |  |  |  |
|         | 100  | 175 | .7 | 77.7 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0)  |
| DIRECT     | The layer direction for BEHi = SLCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer inputs is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer $\pm 1$ , $\pm 2$ , or $\pm 3$ ; Default +1) |
| CORDM      | Identification number of the material coordinate system. See Remark 10. (Integer; Default = 0, which is basic)   |
| SB         | Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real $\geq 0.0$ )   |
| ANAL       | Analysis type. ANAL=‘IS’ - Implicit structural elements are being referred to. ANAL=‘IH’ - Implicit heat analysis elements are being referred to. ANAL=‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH)  |
| C8         | Keyword indicating that two items following apply to elements with eight corner grids. (Character)   |
| C20        | Keyword indicating that two items following apply to elements with eight corner grids and twelve mid-side grids. (Character)   |
| BEHi       | Element structural behavior. See Remarks 4. and 7. (Character default: SLCOMP for BEH8 and BEH20)  |
| INTi       | Integration scheme. See Remark 9. (Character default: L for INT8, Q for INT20)   |

|        |   |
|--------|---|
| BEHiH  | Element heat behavior. See Remarks 4. and 8. (Character Default: SLCOMP for BEH8H and BEH20H)   |
| INTiH  | Integration scheme. See Remark 9. (Character Default: L for INT8H, Q for INT20H)  |
| IDi    | Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)  |
| MIDi   | Material ID for the ply. See Remark 3. (Integer > 0)  |
| Ti     | Either fractional percent of the total element thickness or actual thickness of that ply depending on $\pm$ value of DIRECT. See Remarks 5. and 6.. (Real > 0.0)                        |
| THETAi | Orientation angle of the ply in the plane of the plies. Measured relative to the projection z-axis defined by CORDM on the plane defined by DIRECT. See Remark 1. (Real; Default = 0.0) |

**Remarks:**

1. The PCOMPLS can only be referenced by a CHEXA entry. To view the composite results of large models using solid composite elements in PATRAN or SimXpert, the following environment variable setting is required when starting the session: DRANAS\_NAST\_MEM=2048MB.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across all PCOMPG, PLCOMP, and PCOMPLS entries.
3. The MIDi entry may point to MAT1, MAT9, MATORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

| Implicit Structural Materials                          |         |         |       |        |         |
|--|---------|---------|-------|--------|---------|
| MAT1   | MAT9    | MATORT  | MATHE | MATUSR | MATDIGI |
| MATVE  | <MATVE> | <MATVE> | MATVE |        |         |
| MATVP  | MATVP   | MATVP   |       |        |         |
| MATEP  | MATEP   | MATEP   |       |        |         |
| MATF   | MATF    | MATF    |       |        |         |
| MATS1  |         | MATSORT |       |        |         |
| <MATVE> refers to the ALTERNATE format for type ORTHO. |         |         |       |        |         |

| Heat Materials |      |
|----------------|------|
| MAT4           | MAT5 |

If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, MATG and MATDIGI entries.

If MATDIGI is being used, the ANAL field must be set to IS.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries, between themselves, may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=SLCOMP.

For INT8=L and INT20=Q, a total of 510 layers are allowed for any one element.

For INT8=ASTN, a total of 2040 layers are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

| Layer orientation |                       |   |                     |
|-------------------|-----------------------|---|---------------------|
| DIRECT            | Normal to Layer Plane | Layers run parallel from face (ply numbering starts here) | to face (ends here) |
| 1                 | Element T direction   | G1-G2-G3-G4   | G5-G6-G7-G8         |
| 2                 | Element R direction   | G1-G4-G8-G5   | G2-G3-G7-G6         |
| 3                 | Element S direction   | G1-G2-G6-G5   | G4-G3-G7-G8         |

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and +1. The ply thickness is entered in one of two ways:
  - a. Relative thickness where the numbers entered, are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percents of thickness must sum to 1.0.
  - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plies and then figures the fractional percent of each individual ply as in method 6a.

7. In the following table, BEHi refers to the structural behavior of 3D-solid elements. An underlined item delineates a default.

| Structural Classification of Elements |           |                  |              |         |
|---------------------------------------|-----------|------------------|--------------|---------|
| Element Structural Type               | BEHi CODE | Integration Code | Element Type | # Nodes |
| Solid continuum composite             | SLCOMP    | <u>L</u>         | HEX          | 8       |
|                                       |           | ASTN*            | HEX          | 8       |
|                                       |           | <u>Q</u>         | HEX          | 20      |

\*Only DIRECT=1 or -1 is allowed

8. In the following table, BEHiH refers to the heat behavior of 3D-solid elements. An underlined item delineates a default.

| Heat Classification of Elements |            |                      |              |         |
|---------------------------------|------------|----------------------|--------------|---------|
| Element Heat Type               | BEHiH CODE | Integration Code     | Element Type | # Nodes |
| Solid continuum composite       | SLCOMP     | <u>L</u><br><u>Q</u> | HEX<br>HEX   | 8<br>20 |

9. Integration codes in Remark 7. and 8. are:

| INT CODE | Integration Type                                |
|----------|---|
| L        | Linear  |
| ASTN     | Assumed STrain enhanced formulation solid shell |
| Q        | Quadratic                                       |

10. The material coordinate system CORDM may be the basic system (0 or blank) or the element coordinate system (-1) or any defined coordinate system (> 0) (Default = 0).

THETAi in conjunction with CORDM is used to define the ply orientation.

- a. For BEH = SLCOMP and INT = ASTN, the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETAi is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.
- b. For BEH = SLCOMP and INT = ASTN, the Material Coordinate System is a function of how NLMOPTS, INLAM, CPROJ is set.

If CPROJ is ON then the projection scheme (see Figure #) is activated. With the activation of projection scheme the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETAI is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.

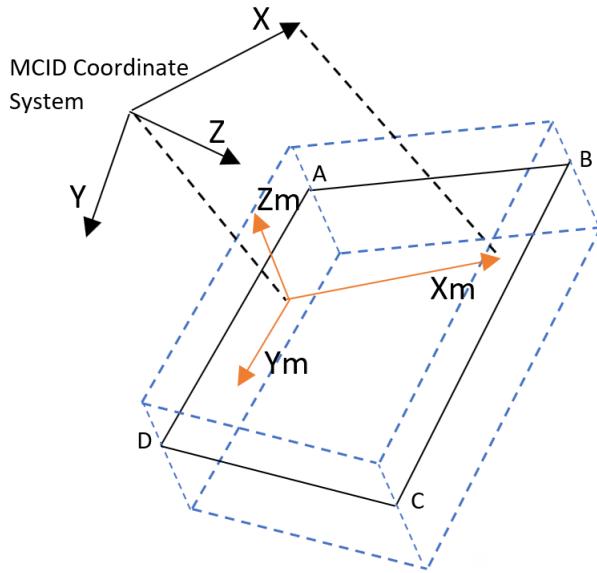


Figure 9-133      Projection Scheme

In Figure 9-133, plane ABCD is the layer plane of interest.  $Z_m$  is normal to the layer plane (or is the thickness direction).  $X_m$  is the projection of the X axis of the MCID coordinate system on the layer plane.  $Z_m \times X_m$  gives the  $Y_m$ . Angle theta is measured counter-clockwise from  $X_m$  axis about the  $Z_m$  axis.

If CPROJ is OFF (default), then the CORDM axes directly form the local X, Y and Z axes of the layer plane. THETAI is then measured positive counter-clockwise about the local Z axis.

11. DIRECT, SB, ANAL, "C8" and "C20" are not supported in SOL 600 will be ignored if entered.
12. For SOL 600, PCOMPLS is made into PSOLID and PCOMP internally. Existing PSOLID and PCOMP entries with the same PID may be overwritten. The combination PSOLID and PCOMP is preferred over PCOMPLS for SOL 600, either option (but not both) may be used.
13. PCOMPLS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPLS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS, PLCOMP, PCOMPG entries.

14. It is a general limitation in SOL 400 analysis = HSTAT and SOL 101 APP HEAT won't support Gradient and Flux calculation in z direction for layered solid composite (material laminate) element.

**PCONEAX**

## Conical Shell Element Property

Defines the properties of a conical shell element described on a CCONEAX entry.

**Format:**

| 1       | 2    | 3    | 4    | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|------|------|------|-------|-------|-------|-------|-------|----|
| PCONEAX | ID   | MID1 | T1   | MID2  | I     | MID3  | T2    | NSM   |    |
|         | Z1   | Z2   | PHI1 | PHI2  | PHI3  | PHI4  | PHI5  | PHI6  |    |
|         | PHI7 | PHI8 | PHI9 | PHI10 | PHI11 | PHI12 | PHI13 | PHI14 |    |

**Example:**

|         |       |        |      |      |      |   |     |     |  |
|---------|-------|--------|------|------|------|---|-----|-----|--|
| PCONEAX | 2     | 4      | 1.0  | 6    | 16.3 | 8 | 2.1 | 0.5 |  |
|         | 0.001 | -0.002 | 23.6 | 42.9 |      |   |     |     |  |

**Descriptor      Meaning**

|        |   |
|--------|---|
| ID     | Property identification number. (Unique Integer > 0)  |
| MIDi   | Material identification number for membrane, bending, and transverse shear. (Integer $\geq 0$ ) |
| T1     | Membrane thickness. (Real > 0.0 if MID1 = 0)  |
| T2     | Transverse shear thickness. (Real > 0.0 if MID3 = 0)  |
| I      | Moment of inertia per unit width. (Real)  |
| NSM    | Nonstructural mass per unit area. (Real)  |
| Z1, Z2 | Fiber distances from the middle surface for stress recovery. (Real)                             |
| PHIi   | Azimuthal coordinates (in degrees) for stress recovery. (Real)                                  |

**Remarks:**

1. PCONEAX is allowed only if an AXIC entry is also present.
2. PCONEAX entries may reference MAT1 or MAT2 material entries. However, only orthotropic material properties are consistent with axisymmetry. Therefore, G13 and G23 values on the MAT2 entry referenced by MID1 or MID2 and the G12 value on the MAT2 entry referenced by MID3 should be set to 0.0. In addition, the MID3 entry, if it references a MAT2 material matrix, should be of size 2 x 2.
3. If either MID1 = 0 or blank or T1 = 0.0 or blank, then both must be zero or blank.
4. If either MID2 = 0 or blank or I = 0.0 or blank, then both must be zero or blank.
5. If either MID3 = 0 or blank or T2 = 0.0 or blank, then both must be zero or blank.
6. A maximum of 14 azimuthal coordinates (PHIi) for stress recovery may be specified.
7. For a discussion of the conical shell problem, see Section 5.3.3 of the *MSC Nastran Reference Guide*.

8. The following elastic relationships are assumed:

- In-plane forces per unit width

$$\{F\} = T1[G_1]\{\varepsilon\}$$

where  $\{\varepsilon\}$  is the vector of strains in the middle surface.

- Bending moments per unit width

$$\{M\} = I[G_2]\{\chi\}$$

where  $\{\chi\}$  is the vector of curvatures.

- Transverse shear forces per unit width

$$\{V\} = T2[G_3]\{\gamma\}$$

where  $\{\gamma\}$  is the vector of transverse shear strains.

$[G_1]$ ,  $[G_2]$  and  $[G_3]$  are the stress-strain matrices defined by MID1, MID2, and MID3, respectively.

**Pconv****Convection Property Definition**

Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

**Format:**

| 1     | 2      | 3     | 4    | 5    | 6     | 7   | 8 | 9 | 10 |
|-------|--------|-------|------|------|-------|-----|---|---|----|
| Pconv | Pconid | MID   | Form | Expf | Ftype | Tid |   |   |    |
|       | Chlen  | Gidin | Ce   | E1   | E2    | E3  |   |   |    |

**Examples:**

|       |     |     |   |     |     |     |  |  |  |
|-------|-----|-----|---|-----|-----|-----|--|--|--|
| Pconv | 53  | 2   | 0 | .25 |     |     |  |  |  |
| Pconv | 4   |     |   |     | 1   | 101 |  |  |  |
| Pconv | 38  | 21  |   |     | 2   | 54  |  |  |  |
|       | 2.0 | 235 | 0 | 1.0 | 0.0 | 0.0 |  |  |  |

**Alternate Format and Examples:**

| 1     | 2      | 3   | 4    | 5    | 6   | 7     | 8     | 9     | 10 |
|-------|--------|-----|------|------|-----|-------|-------|-------|----|
| Pconv | Pconid | MID | Form | Expf | "3" | H1    | H2    | H3    |    |
|       | H4     | H5  | H6   | H7   | H8  |       |       |       |    |
| Pconv | 20     |     |      |      | 3   | 10.0  |       |       |    |
| Pconv | 7      |     |      |      | 3   | 10.32 | 10.05 | 10.09 |    |
|       | 10.37  |     |      |      |     |       |       |       |    |

| Descriptor | Meaning   |
|------------|---|
| Pconid     | Convection property identification number. (Integer > 0)  |
| MID        | Material property identification number. See Remarks 2. (Integer > 0)   |
| Form       | Type of formula used for free convection. (Integer 0, 1, 10, 11, 20, or 21; Default = 0)  |
| Expf       | Free convection exponent as implemented within the context of the particular form that is chosen. See Remark 3. (Real $\geq 0.0$ ; Default = 0.0)                                     |
| Ftype      | Formula type for various configurations of free convection. See Remarks 2. and 5. (Integer $\geq 0$ ; Default = 0)  |
| Tid        | Identification number of a TABLEHT entry that specifies the two-variable tabular function of the free convection heat transfer coefficient. See Remark 5. (Integer $\geq 0$ or blank) |
| Chlen      | Characteristic length. See Remarks 6. and 8. (Real $> 0.0$ or blank)  |

| Descriptor | Meaning   |
|------------|---|
| GIDIN      | Grid ID of the referenced inlet point. See Remarks 7. and 8. (Integer > 0 or blank)   |
| CE         | Coordinate system for defining the direction of boundary-layer flow. See Remarks 7. and 8. (Integer $\geq 0$ ; Default = 0)                 |
| Ei         | Component of the vector for defining the direction of boundary-layer flow in coordinate system CE. See Remarks 7. and 8. (Real or blank)    |
| Hi         | Free convection heat transfer coefficient. See Remark 5. (Real for H1 and Real or blank for H2 through H8; Default for H2 through H8 is H1) |

### Remarks:

- Every surface to which free convection is to be applied must reference a PCONV entry. PCONV is referenced on the CONV Bulk Data entry.
- MID is used to supply the convection heat transfer coefficient (H) for FTYPE=0, or the thermal conductivity (K) for FTYPE=2. MID is ignored for FTYPE=1 and FTYPE=3 and may be blank.
- EXPF is the free convection temperature exponent.
  - If FORM = 0, 10, or 20, EXPF is an exponent of  $(T - TAMB)$ , where the convective heat transfer is represented as
 
$$q = H \cdot u_{CNTRLND} \cdot (T - TAMB)^{EXPF} \cdot (T - TAMB).$$
  - If FORM = 1, 11, or 21,
 
$$q = H \cdot u_{CNTRLND} \cdot (T^{EXPF} - TAMB^{EXPF})$$

where T represents the elemental grid point temperatures and TAMB is the associated ambient temperature.
- FORM specifies the formula type and the reference temperature location used in calculating the convection film coefficient if FLMND = 0.
  - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperatures (average).
  - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
  - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperatures).
- FTYPE defines the formula type used in computing the convection heat transfer coefficient h.
  - If FTYPE = 0, h is specified in the MAT4 Bulk Data entry referenced by MID.
  - If FTYPE = 1, h is computed from  $h = f(T_w, T_a)$ , where f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referenced by TID,  $T_w$  is the wall temperature, and  $T_a$  is the ambient temperature.
  - If FTYPE = 2, h is computed from  $Nu = f(T_w, T_a)$ , where  $Nu_L = hL/K$  or  $Nu_x = hX/K$  is the Nusselt number, f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referred by TID,  $T_w$  is the wall temperature, and  $T_a$  is the ambient temperature.

- If FTYPE = 3,  $h$  is the free convection heat transfer coefficient applied to grid point Gi of the referenced HBDY surface element.
6. CHLEN specifies the characteristic length used to compute the average heat transfer coefficient  $\bar{h}$ . The following table lists typical values of CHLEN for various convection configurations.

| Convection Configuration                           | Characteristic Length CHLEN   |
|--|---|
| Free convection on a vertical plate or cylinder    | Height of the plate or cylinder   |
| Free convection from horizontal tubes              | Diameter of the pipes   |
| Free convection from horizontal square plates      | Length of a side  |
| Free convection from horizontal rectangular plates | Average length of four sides  |
| Free convection from horizontal circular disks     | 0.9d, where d is the diameter of the disk.                              |
| Free convection from horizontal unsymmetric plates | A/P, where A is the surface area and P is the perimeter of the surface. |

7. GIDIN, CE and Ei are used to define the distance from the leading edge of heat transfer. GIDIN specifies the referenced grid ID where heat transfer starts. CE and Ei define the direction of boundary-layer flow. If CE field is blank, the default is CE=0 for basic coordinate system. If E1, E2, and E3 fields are blank, the defaults are Ei = < 1.0, 0.0, 0.0 >, i.e. the flow is in the x direction.
8. CHLEN, GIDIN, CE, and Ei are required only for free convection from flat plates with FTYPE = 2. In this case, if the heat transfer coefficient is spatial dependent, GIDIN must be specified. Otherwise, CHLEN has to be defined for the computation of average heat transfer coefficient  $\bar{h}$ . For free convection from tubes (CHBDYP elements with TYPE="ELCY", "TUBE" or "FTUBE"), CHLEN, GIDIN, CE, and Ei need not be specified, because MSC Nastran will use the average diameter of tubes as the characteristic length while computing Nu. CHLEN, GIDIN, CE, and Ei are ignored for  $FTYPE \neq 2$ .
9. For RC network solver in thermal analysis, the FORM, EXPF, FTYPE, TID, CHLEN, GIDIN, CE, E1, E2 and E3 are ignored.

**PCONVM**

## Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

**Format:**

| 1      | 2      | 3   | 4    | 5    | 6    | 7    | 8     | 9    | 10 |
|--------|--------|-----|------|------|------|------|-------|------|----|
| PCONVM | PCONID | MID | FORM | FLAG | COEF | EXPR | EXPPI | EXPO |    |

**Example:**

|        |   |   |   |   |      |      |      |      |  |
|--------|---|---|---|---|------|------|------|------|--|
| PCONVM | 3 | 2 | 1 | 1 | .023 | 0.80 | 0.40 | 0.30 |  |
|--------|---|---|---|---|------|------|------|------|--|

**Descriptor      Meaning**

|        |   |
|--------|---|
| PCONID | Convection property identification number. (Integer > 0)  |
| MID    | Material property identification number. (Integer > 0)  |
| FORM   | Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)                           |
| FLAG   | Flag for mass flow convection. (Integer = 0 or 1; Default = 0)  |
| COEF   | Constant coefficient used for forced convection. (Real > 0.0)   |
| EXPR   | Reynolds number convection exponent. (Real $\geq$ 0.0; Default = 0.0)   |
| EXPPI  | Prandtl number convection exponent for heat transfer into the working fluid. (Real $\geq$ 0.0; Default = 0.0)   |
| EXPO   | Prandtl number convection exponent for heat transfer out of the working fluid. (Real $\geq$ 0.0; Default = 0.0) |

**Remarks:**

1. Every surface to which forced convection is applied must reference a PCONVM entry. PCONVM is referenced on the CONVM entry.
2. MID specifies material properties of the working fluid at the temperature of the point FLMND. FLMND is specified on the CONVM entry.
3. The material properties are used in conjunction with the average diameter and mass flow rate (mdot). MID references the material properties and supplies the fluid conductivity (k), heat capacity (cp), and viscosity ( $\mu$ ) needed to compute the Reynolds (Re) and Prandtl (Pr) numbers as follows:

$$\text{Re} = 4 \cdot |\text{mdot}| / (\pi \cdot \text{diameter} \cdot \mu)$$

$$\text{Pr} = \text{cp} \cdot \mu / k$$

4. FORM controls the type of formula used in determination of the forced convection film coefficient h. There are two cases:
  - If FORM = 0, 10, or 20 than  $h = \text{coef} \cdot \text{Re}^{\text{EXPR}} \cdot \text{Pr}^{\text{EXPPI}}$ .

- If FORM = 1, 11, or 21 then the above  $h$  is multiplied by  $k$  and divided by the average hydraulic diameter.
  - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.
  - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).
  - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
  - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).
5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.
  6. FLAG controls the convective heat transfer into the downstream point (the second point as identified on the CHBDYi statement is downstream if mdot is positive).
    - FLAG = 0, no convective flow (stationary fluid).
    - FLAG = 1, convective energy flow that is consistent with the Streamwise Upwind Petrov Galerkin (SUPG) element formulation.
  7. No phase change or internal heat generation capabilities exist for this element.
  8. For RC network solver in thermal analysis, the FORM, FLAG, COEF, EXPR, EXPPI and EXRPO are ignored.

**PCONV1****Thermal Convection Calculation Properties**

Defines the properties required to calculate convective heat transfer. Can exist in a simple mode with convection coefficient defined in the MID or in advanced mode where the H value is calculated using the geometric parameters and referenced material.

**Format:**

| 1      | 2   | 3   | 4                    | 5                   | 6      | 7          | 8                    | 9                  | 10 |
|--------|-----|-----|----------------------|---------------------|--------|------------|----------------------|--------------------|----|
| PCONV1 | PID |     | CorrID               | MID                 | Mdot   | Velocity   | Length or Diameter   | Flow Cross Section |    |
|        |     |     | Length function type | Flow Cross Sec type | Mdot f | Velocity f | Length or Diameter f | Flow Cross Sec f   |    |
|        | C1  | C2  | C3                   | C4                  | C5     | C6         | C7                   | C8                 |    |
|        | C9  | C10 | C11                  | C12                 | C13    | C14        | C15                  | C16                |    |
|        | C17 | C18 | C19                  | C20                 | C21    | C22        | C23                  | C24                |    |

**Example:**

|        |   |  |     |   |  |  |     |  |  |
|--------|---|--|-----|---|--|--|-----|--|--|
| PCONV1 | 2 |  | 701 | 2 |  |  | 1.0 |  |  |
|--------|---|--|-----|---|--|--|-----|--|--|

| Descriptor           | Meaning  |
|----------------------|--|
| PID                  | Property identification number, referenced by CHBDYP and/or CONV. (Integer > 0)        |
| Corr ID              | ID of convection type in correlation library (Integer $\geq 0$ ; Default 0)            |
| MID                  | Material identification number for convection calculation (Integer > 0)                |
| Mdot                 | Mass flow rate (Real; Default 0.0)   |
| Velocity             | Fluid velocity (Real $\geq 0.0$ ; Default 0.0)   |
| Length or Diameter   | Geometry, based on Corr ID (Real $\geq 0.0$ ; Default 0.0)                             |
| Flow Cross Section   | Flow Cross Section (Real $\geq 0.0$ ; Default 0.0)                                     |
| Length function type | Geometry function type, 1=time 2=temperature (Integer > 0; Default 2)                  |
| Flow Cross Sec type  | Flow Cross Section function type, 1=time 2=temperature (Integer > 0; Default 2)        |
| Mdot f               | Mass flow rate TABLEDj table ID (Integer $\geq 0$ ; Default 0)                         |
| Velocity f           | Fluid velocity TABLEDj table ID (Integer $\geq 0$ ; Default 0)                         |
| Length or Diameter f | Geometric function TABLEMj or TABLEDj table ID (Integer $\geq 0$ ; Default 0)          |
| Flow Cross Sec f     | Flow Cross Section function TABLEMj or TABLEDj table ID (Integer $\geq 0$ ; Default 0) |
| C1 thru C24          | Geometric properties based on chosen convection correlation (Real; Default 0.0)        |

### Remarks:

1. This entry is for RC Network solver only.
2. The PCONV1 entry contains the properties for a CONV and CHDBYP, and can be used for connecting with a PRJCON. PID must be unique to both the PCONVID in PCONV and the PID in PHBDY. This will be the ID referenced by CONV, PRJCON, and CHBDYP.
3. MATID must reference a MAT4 fluid material.
4. For Corr. ID and C1 thru C24, please reference *MSC SINDA User's Guide and Library Reference* or *P/Thermal User's Guide*.

**PDAMP****Scalar Damper Property**

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

**Format:**

| 1     | 2    | 3  | 4    | 5  | 6    | 7  | 8    | 9  | 10 |
|-------|------|----|------|----|------|----|------|----|----|
| PDAMP | PID1 | B1 | PID2 | B2 | PID3 | B3 | PID4 | B4 |    |

**Example:**

|       |    |     |   |     |  |  |  |  |  |
|-------|----|-----|---|-----|--|--|--|--|--|
| PDAMP | 14 | 2.3 | 2 | 6.1 |  |  |  |  |  |
|-------|----|-----|---|-----|--|--|--|--|--|

**Describer      Meaning**

PIDI              Property identification number. (Integer > 0)

Bi              Force per unit velocity. (Real)

**Remarks:**

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. PDAMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PDAMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PDAMP entries.

**PDAMP5**

## Scalar Damper Property for CDAMP5

Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.

**Format:**

| 1      | 2   | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|---|---|---|---|---|---|----|
| PDAMP5 | PID | MID | B |   |   |   |   |   |    |

**Example:**

|        |   |   |     |  |  |  |  |  |  |
|--------|---|---|-----|--|--|--|--|--|--|
| PDAMP5 | 2 | 3 | 4.0 |  |  |  |  |  |  |
|--------|---|---|-----|--|--|--|--|--|--|

**Describer      Meaning**

- PID      Property identification number. (Integer > 0)  
MID      Material identification number of a MAT4 or MAT5 entry. (Integer > 0)  
B      Damping multiplier. (Real > 0.0)

**Remarks:**

1. B is the mass that multiplies the heat capacity CP on the MAT4 or MAT5 entry.
2. RC network solver does not support PDAMP5 for thermal analysis.

**PDAMPT**

## Frequency-Dependent Damper Property

Defines the frequency-dependent properties for a PDAMP Bulk Data entry.

**Format:**

| 1      | 2    | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|------|-------|---|---|---|---|---|---|----|
| PDAMPT | PID1 | TBID1 |   |   |   |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PDAMPT | 12 | 34 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

**Descriptor      Meaning**

- PID      Property identification number that matches the identification number on a PDAMP entry. (Integer > 0)
- TBID1      Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship. (Integer  $\geq 0$ ; Default = 0)

**Remarks:**

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements in the residual structure, which do not attach to any omitted degrees-of-freedom.
2. The PDAMPT entry is ignored in all solution sequences except frequency response analysis.

**PDISTB**

Element property distributions.

Defines element distributions of property data.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6       | 7      | 8     | 9  | 10 |
|--------|--------|--------|--------|--------|---------|--------|-------|----|----|
| PDISTB | ID     | TYPE   | PID    | PNAME  | LOCTNUM | SMULT  | DELTA |    |    |
|        | EID1   | V1     | EID2   | V2     | EID3    | V3     | EID4  | V4 |    |
|        | EID5   | V5     | EID6   | V6     | EID7    | V7     | EID8  | V8 |    |
|        | -etc.- | -etc.- | -etc.- | -etc.- | -etc.-  | -etc.- |       |    |    |

**Example:**

|        |       |            |     |      |     |      |     |      |  |
|--------|-------|------------|-----|------|-----|------|-----|------|--|
| PDISTB | 7008  | PCOMP<br>G | 700 | GE   |     | .9   |     |      |  |
|        | 205   | .013       | 224 | .015 | 362 | .135 |     |      |  |
| PDISTB | 70014 | PCOMP<br>G | 700 | T    | 1   |      |     |      |  |
|        | 205   | .013       | 224 | .015 | 362 | .135 |     |      |  |
| PDISTB | 70034 | PCOMP<br>G | 700 | T    | 3   |      |     |      |  |
|        | 205   | .013       | 224 | .015 | 362 | .135 | 400 | .067 |  |

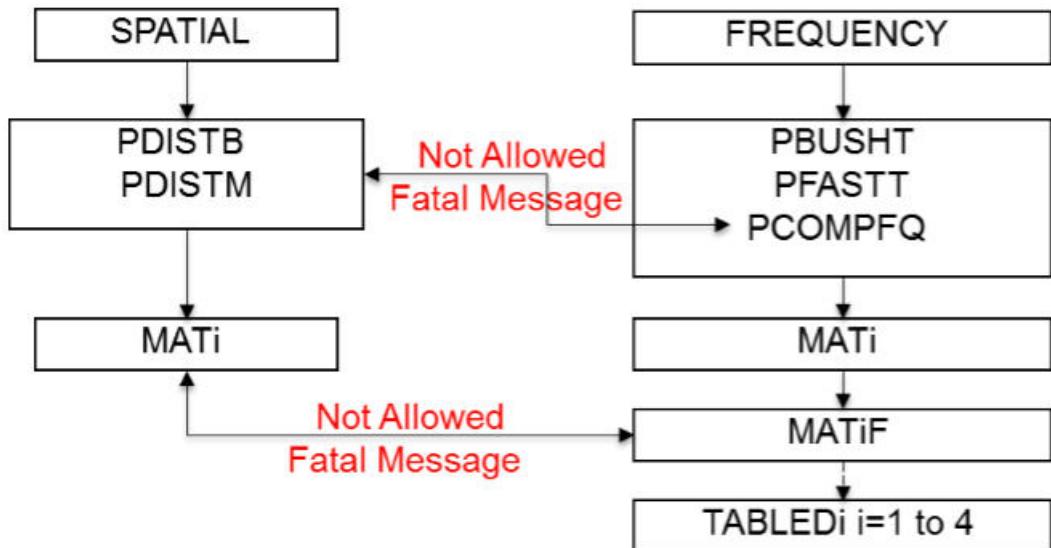
**Descriptor      Meaning**

- ID              Unique identification number. (Integer > 0, no default)
- TYPE          Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL and PTUBE (Character-Required).
- PID          Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default)
- PNAME        Property name, such as "T", "I12" (Character). See Remark 8..
- LOCNUM       The Numerical order of Ply in PCOMP or PCOMPG. The Station number on a PBEAM or PBEAM3. (Integer ≥ 0).
- SMULT        Scale factor of distribution entries. (Real, Default=1.0)
- DELTA        Delta value of distribution entries. (Real, Default=0.0)
- EIDi         Element ID associated with Property PID. (Integer > 0, no default)
- Vi            The value of item to be changed. (Real)

**Remarks:**

1. The ID of the PDISTB is for user convenience of grouping identification and is not used for case control selection.

2. The PID must point to an existing “PSHELL”, “PCOMPG”, and “PCOMP” entry. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
3. If no distribution is given, the value associated with the named property entry will be used.
4. If an EID occurs on this entry for an element not associated with referred to property the EID is ignored.
5. If an EID is referred to twice on this entry or another PDISTB entry with the same PID and PNAME, Or in the case of a PCOMP or PCOMPG with same PID, PNAME, and LOCNUM referral a fatal message will be issued.
6.  $\text{VALUE}_i = \text{SMULT}^* V_i + \text{DELTA}$
7. The PID referred to may not have an associated PCOMPFQ entries. A fatal message will be issued if there is such an association. Additionally a fatal message will be issued if the associated materials are associated with a MATiF entry.



8. For most Primary property entries, the QRG symbol is used for valid PNAME. See values in table below. For more difficult entries such as a PCOMP the PNAME for example of the thickness of PLY 3 would be entered as PNAME=T and LOCTNUM=3. Note on PCOMPG it is the numerical PLY listed not the GPLYID. Thus for example if the second ply listed on a PCOMPG entry had GPLYID=4 and the thickness is to be changed then PNAME=T and LOCTNUM=2 not 4.

| Property                     | PNAME       | Corresponding QRG Description  |
|------------------------------|-------------|--|
| Current Supported Items are: |             |  |
| PSHELL                       | T           | T - Membrane thickness   |
|                              | I12         | $12I/T^{**3}$ Bending moment of inertia ratio  |
|                              | TS          | TS/T Transverse shear thickness ratio  |
|                              | NSM         | Nonstructural mass.  |
|                              | Z1, Z2      | Fiber distances for stress calculations  |
| PCOMP (G)                    | Z0          | Distance from the reference plane to the bottom surface  |
|                              | NSM         | Nonstructural mass.  |
|                              | SB          | Allowable shear stress of the bonding material   |
|                              | TREF        | Reference temperature  |
|                              | GE          | Damping coefficient  |
|                              | T           | Thicknesses of the various plies (Ply Identified by LOCTNUM)   |
|                              | THETA       | Orientation angle of the various plies. (Ply Identified by LOCTNUM)                                      |
| PBEAM                        | LOCTNUM=1   | corresponds to END A; LOCTNUM=12 corresponds to END B; LOCTNUM=2-11 Intermediate stations left to right. |
|                              | A           | Area of bar cross section (Identified by LOCTNUM)  |
|                              | I1, I2, I12 | Area moments of inertia (Identified by LOCTNUM)  |
|                              | J           | Torsional constant (Identified by LOCTNUM)   |
|                              | NSM         | Nonstructural mass. (Identified by LOCTNUM)  |
|                              | K1, K2      | Shear stiffness factor K in $K^*A^*G$ for plane 1 and plane 2.   |
| PROD                         | A           | Area of bar cross section  |
|                              | J           | Torsional constant   |
|                              | NSM         | Nonstructural mass.  |
| PBAR                         | A           | Area of bar cross section  |
|                              | I1, I2, I12 | Area moments of inertia  |
|                              | J           | Torsional constant   |
|                              | NSM         | Nonstructural mass.  |
|                              | K1, K2      | Area factor for shear  |

| Property                     | PNAME       | Corresponding QRG Description  |
|------------------------------|-------------|--|
| Current Supported Items are: |             |  |
| PTUBE                        | OD          | Outside diameter of tube   |
|                              | T           | Thickness of tube  |
|                              | NSM         | Nonstructural mass.  |
|                              | OD2         | Diameter of tube at second grid point  |
| PSHEAR                       | T           | Thickness of shear panel.  |
|                              | NSM         | Nonstructural mass.  |
|                              | F1, F2      | Effectiveness factor for extensional stiffness along edges 2-3 and 1-4.          |
| PBEAM3                       | LOCNUM=1    | corresponds to END A; LOCNUM=2 corresponds to END B; LOCNUM=3 “middle” station C |
|                              | A           | Area of the beam cross-section (Identified by LOCNUM)                            |
|                              | IZ, IY, IYZ | Area moments of inertia (Identified by LOCNUM)                                   |
|                              | J           | Torsional constant (Identified by LOCNUM)  |
|                              | NSM         | Nonstructural mass. (Identified by LOCNUM)                                       |
|                              | KY, KZ      | Shear effectiveness factors for local y- and z-directions.                       |
|                              | CW          | Warping coefficient (Identified by LOCNUM).                                      |

**PDISTBM**

Element property material distributions.

Defines element distributions of property data.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8    | 9     | 10 |
|---------|--------|--------|--------|--------|--------|--------|------|-------|----|
| PDISTBM | ID     | TYPE   | PID    | MIDNUM |        |        |      |       |    |
|         | EID1   | MIDV1  | EID2   | MIDV2  | EID3   | MIDV3  | EID4 | MIDV4 |    |
|         | EID5   | MIDV5  | EID6   | MIDV6  | EID7   | MIDV7  | EID8 | MIDV8 |    |
|         | -etc.- | -etc.- | -etc.- | -etc.- | -etc.- | -etc.- |      |       |    |

**Example:**

|         |       |        |     |   |  |  |  |  |  |
|---------|-------|--------|-----|---|--|--|--|--|--|
| PDISTBM | 70013 | PSHELL | 700 | 2 |  |  |  |  |  |
|         | 205   | 16     |     |   |  |  |  |  |  |

**Descriptor      Meaning**

|        |  |
|--------|--|
| ID     | Unique identification number. (Integer > 0, no default)  |
| TYPE   | Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL, PSOLID, and PTUBE (Character-Required).    |
| PID    | Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default) |
| MIDNUM | MIDI name (Integer $\geq 0$ , Default=1) See Remark 6.   |
| EIDi   | Element ID associated with Property PID. (Integer > 0, no default)   |
| MIDVi  | The replacement value of the MIDi. (Integer>0, no default)   |

**Remarks:**

1. The PID must point to an existing “PSHELL”, “PCOMPG”, “PCOMP”, and “PSOLID”, entry. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
2. If no distribution is given, the value associated with the named property entry will be used.
3. If an EID occurs on this entry for an element not associated with referred to property entry the EID is ignored.
4. If an EID is referred to twice on this entry or another PDISTBM entry with the same PID and MIDNUM referral a fatal message will be issued.
5. The MIDVi the user supplies must be a valid MATi entry unique across all existing MATi entries. The MATi referred to may not have associated MATiF entries. A fatal message will be issued if there is such an association.

6. The MIDNUM is property dependent. For example a PSHELL has MID1 through MID4, so a MIDNUM=3 means that the MID3 field of the PSHELL is being changed. For PCOMP or PCOMPG the MIDNUM refers to the MIDi of the Numerical order of ply.

**PDUMi****Dummy Element Property**

Defines the properties of a dummy element ( $3 \leq i \leq 7$ ). Referenced by the CDUMi entry.

**Format:**

| 1     | 2   | 3      | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|-----|--------|----|----|----|----|----|----|----|
| PDUMi | PID | MID    | A1 | A2 | A3 | A4 | A5 | A6 |    |
|       | A7  | -etc.- |    |    |    |    |    |    |    |

**Example:**

|       |     |   |     |     |      |     |  |     |  |
|-------|-----|---|-----|-----|------|-----|--|-----|--|
| PDUM3 | 108 | 2 | 2.4 | 9.6 | 1.E4 | 15. |  | 3.5 |  |
|       | 5   |   | 2   |     |      |     |  |     |  |

**Descriptor      Meaning**

- |     |   |
|-----|---|
| PID | Property identification number. (Integer > 0) |
| MID | Material identification number. (Integer > 0) |
| Aj  | Additional fields. (Real or Integer)          |

**Remark:**

1. The additional fields are defined in the user-written element subroutines.
2. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran". Other than the PID field, all field checking is the responsibility of the user supplied code.

**PELAS****Scalar Elastic Property**

Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).

**Format:**

| 1     | 2    | 3  | 4   | 5  | 6    | 7  | 8   | 9  | 10 |
|-------|------|----|-----|----|------|----|-----|----|----|
| PELAS | PID1 | K1 | GE1 | S1 | PID2 | K2 | GE2 | S2 |    |

**Example:**

|       |   |      |      |      |    |      |        |  |  |
|-------|---|------|------|------|----|------|--------|--|--|
| PELAS | 7 | 4.29 | 0.06 | 7.92 | 27 | 2.17 | 0.0032 |  |  |
|-------|---|------|------|------|----|------|--------|--|--|

**Descriptor      Meaning**

|      |   |
|------|---|
| PIDI | Property identification number. (Integer > 0)             |
| Ki   | Elastic property value. (Real)                            |
| GEi  | Damping coefficient, $g_e$ . See Remarks 5. and 6. (Real) |
| Si   | Stress coefficient. (Real)                                |

**Remarks:**

1. Be careful using negative spring values.
2. Spring values are defined directly on the CELAS2 and CELAS4 entries, and therefore do not require a PELAS entry.
3. One or two elastic spring properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. If PARAM,W4 is not specified, GEi is ignored in transient analysis. See [Parameters, 783](#).
6. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
7. If PELAS is used in conjunction with PELAST, Ki > 0, and the initial slope of the nonlinear force-displacement relationship defined by the PELAST should agree with Ki.
8. PELAS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PELAS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PELAS entries.

## PELAS1

Defines a spring property designated by a force-deflection curve for SOL 700. Used in SOL 700 only.

### Format:

| 1      | 2   | 3   | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|---|---|---|---|---|---|----|
| PELAS1 | PID | TID |   |   |   |   |   |   |    |

### Example:

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PELAS1 | 22 | 33 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

PID              Property identification number. (Integer, no Default, > 0)

TID              Identification number of a TABLED1 entry which defines the force deflection curve.  
(Integer, no Default, > 0)

### Remarks:

1. Unlike PELAST, when PELAS1 is used, no PELAS entry is made.
2. All PELAS and PELAS1 ID's must be unique.
3. This entry may only be referenced by a CELAS1D entry.

**PELAST**

## Frequency Dependent or Nonlinear Elastic Property

Defines the frequency dependent or nonlinear properties for a PELAS Bulk Data entry.

**Format:**

| 1      | 2   | 3    | 4     | 5     | 6 | 7 | 8 | 9 | 10 |
|--------|-----|------|-------|-------|---|---|---|---|----|
| PELAST | PID | TKID | TGEID | TKNID |   |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PELAST | 44 | 38 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

**Describer      Meaning**

- |       |  |
|-------|--|
| PID   | Property identification number that matches the identification number on a PELAS entry. (Integer > 0)  |
| TKID  | Identification number of a TABLEDi entry that defines the force per unit displacement vs. frequency relationship. (Integer > 0; Default = 0)                   |
| TGEID | Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0) |
| TKNID | Identification number of a TABLEDi entry that defines the nonlinear force vs. displacement relationship. (Integer > 0; Default = 0)                            |

**Remarks:**

1. The PELAST entry may only be referenced by CELAS1 or CELAS3 elements in the residual structure which do not attach to any omitted degrees-of-freedom.
2. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PELAS entry.
3. The nominal values are used for all analysis types except frequency response and nonlinear analyses. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. For nonlinear analysis the nominal values for Ki should agree with the initial slope of the nonlinear force-displacement relationship defined by the PELAST, or the results will be unpredictable.

4. The following table summarizes the usage PELAST entry in various solution sequences

| Field | Frequency Response | NONLINEAR (See Remark 6) | Linear (Non-Frequency Response) |
|-------|--------------------|--------------------------|---------------------------------|
| TKID  | Used               | Ignored                  | Ignored                         |
| TGEID | Used               | Ignored                  | Ignored                         |
| TKNID | Ignored            | Used                     | Ignored                         |

5. PELAST is used in SOL 108 and SOL 111 when TKID or TGEID is specified. It is used in SOL 106, SOL 129 and SOL 400 when TKIND is specified. It is ignored in all other solution sequences.
6. This entry is not available in SOL 600 and if entered will cause the job to terminate. If PARAM,MRPELAST,1 is entered this entry will be ignored in SOL 600.

**PERMEAB**

## Air Bag Permeability

Defines the permeability of a COUPLE and/or GBAG (sub)surface.

Permeability is the velocity of gasflow through a (sub)surface and is defined as a linear or tabular function of the pressure difference over the surface. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5    | 6    | 7      | 8      | 9  | 10 |
|---------|-----|-------|-------|------|------|--------|--------|----|----|
| PERMEAB | PID | PERMC | PERMT | FLOW | PENV | RHOENV | SIEENV | CP |    |

**Example:**

|         |     |     |  |     |      |       |        |       |  |
|---------|-----|-----|--|-----|------|-------|--------|-------|--|
| PERMEAB | 201 | 0.5 |  | OUT | 1.E5 | 1.128 | 2.21E5 | 1001. |  |
|---------|-----|-----|--|-----|------|-------|--------|-------|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Unique identification number of a PERMEAB entry. (Integer > 0, Required)   |
| PERMC      | Permeability is a linear function of the pressure difference.<br>permeability = PERM – C*abs (Pinside – PENV)<br>For Pinside > PENV: outflow<br>For Pinside < PENV: inflow<br>See Remark 3. (Real $\geq 0$ )             |
| PERMT      | Permeability is a tabular function of the pressure difference:<br>table contains: permeability versus  Pinside – PENV <br>For Pinside > PENV: outflow<br>For Pinside < PENV: inflow<br>See Remark 3. (Integer $\geq 0$ ) |
| FLOW       | Defines the allowed directions of the flow. (Character, Default = BOTH)<br>BOTH In- and outflow are allowed.<br>IN Only inflow allowed.<br>OUT Only outflow allowed.   |
| PENV       | Environmental pressure. (Real > 0, Required)   |
| RHOENV     | Environmental density. (Real > 0, Required)  |
| SIEENV     | Environmental specific internal energy. (Real > 0, Required)   |
| CP         | Environmental specific heat at constant pressure. See Remark 5. (Real > 0)   |

**Remarks:**

1. The PERMEAB entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver or Roe solver using an EOSGAM (ideal gas) equation of state.

3. Either PERM-C or PERM-T must be specified.
4. The values for the environment  $p_{env}$  (PENV),  $\rho_{env}$  (RHOENV),  $e_{env}$  (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The  $\gamma_{env}$  is calculated and is used when inflow occurs. Inflow occurs when  $p_{env} > p_{inside}$ .

5. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.

**PERMGBG**

## Air Bag Permeability

Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.

The velocity of the gas flow through the surface is defined as a linear or tabular function of the pressure difference. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5    | 6    | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|------|------|---|---|---|----|
| PERMGBG | PID | PERMC | PERMT | FLOW | GBID |   |   |   |    |

**Example:**

|         |    |  |    |  |   |  |  |  |  |
|---------|----|--|----|--|---|--|--|--|--|
| PERMGBG | 12 |  | 10 |  | 2 |  |  |  |  |
|---------|----|--|----|--|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Unique identification number of a PERMEAB entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or from a LEAKAGE to model the flow between an Eulerian air bag and a GBAG. (Integer > 0; Required)  |
| PERMC      | Permeability is a linear function of the pressure difference.<br>permeability = PERM – C*abs (Pinside – Pgbid)<br>The gas flow is from the higher to the lower pressure.<br>See Remark 3. (Real > 0)  |
| PERMT      | Permeability is a tabular function of the pressure difference:<br>table contains: permeability versus  Pinside – Pgbid <br>The gas flow is from the higher to the lower pressure.<br>See Remark 3. (Integer > 0)  |
| FLOW       | Defines the allowed directions of the flow. (Character; Default = BOTH)<br><br>BOTH      In- and outflow are allowed.<br><br>IN          Only inflow allowed into the GBAG or the coupling surface that references this entry.<br><br>OUT        Only outflow allowed into the GBAG or the coupling surface that references this entry. |
| GBID       | Number of a GBAG entry.<br>This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)   |

**Remarks:**

1. The PERMGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.

3. Either PERMC or PERMT must be specified.

**PEULER**

## Eulerian Element Properties

Defines the properties of Eulerian element. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3   | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|------|---|---|---|---|---|----|
| PEULER | PID | MID | TYPE |   |   |   |   |   |    |

**Example:**

|        |     |    |  |  |  |  |  |  |  |
|--------|-----|----|--|--|--|--|--|--|--|
| PEULER | 100 | 25 |  |  |  |  |  |  |  |
|--------|-----|----|--|--|--|--|--|--|--|

**Describer      Meaning**

|      |  |  |  |  |  |  |  |  |  |
|------|--|--|--|--|--|--|--|--|--|
| PID  | Unique property number. (Integer > 0, Required)  |  |  |  |  |  |  |  |  |
| MID  | Number of a MATDEUL entry defining the constitutive model. (Integer $\geq 0$ , Required)                                     |  |  |  |  |  |  |  |  |
| TYPE | The type of Eulerian material being used. (Character, Default = HYDRO)   |  |  |  |  |  |  |  |  |
|      | HYDRO                    Hydrodynamic material with no shear strength + void.  |  |  |  |  |  |  |  |  |
|      | 1STORDER                Single material, 1 <sup>st</sup> order accurate Riemann solution-based fluids- & gases Euler solver. |  |  |  |  |  |  |  |  |
|      | 2NDORDER                Single material, 2 <sup>nd</sup> order accurate Riemann solution-based fluids- & gases Euler solver. |  |  |  |  |  |  |  |  |
|      | STRENGTH                Structural material with shear strength + void.  |  |  |  |  |  |  |  |  |
|      | MMHYDRO                Multimaterial hydrodynamic material with no shear strength + void.                                    |  |  |  |  |  |  |  |  |
|      | MMSTREN                Structural multimaterial with shear strength + void.  |  |  |  |  |  |  |  |  |

**Remarks:**

1. Make the property number unique with respect to all other property numbers.
2. The elements that reference this property use the Eulerian formulation.
3. If TYPE is set to HYDRO, only one material number for all the Eulerian elements of TYPE is used and a hydrodynamic yield model is chosen.
4. If the TYPE is set to either 1st Order or 2nd Order, only one material for all Eulerian elements of TYPE is used and the Riemann solution-based solver is chosen.
5. If TYPE is set to STRENGTH, only one material number for all the Eulerian elements of TYPE is used and a nonhydrodynamic yield model is chosen.
6. If TYPE is set to MMHYDRO, different material numbers for all Eulerian elements of TYPE are used and a hydrodynamic behavior is chosen for each material.
7. If TYPE is set to MMSTREN, different material numbers for all Eulerian elements of TYPE are used and a yield model is chosen for each material.

8. In a multimaterial Euler calculation, the options MMSTREN and MMHYDRO cannot be mixed; they are mutually exclusive.
9. If the material number is blank or zero, the corresponding elements are void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1st Order or 2nd Order scheme, an error message will be issued and the analysis will stop.
10. Initial conditions are defined on the TICEL Bulk Data entry.

**PEULER1**

## Eulerian Element Properties

Eulerian element properties. The initial conditions of these elements are defined in geometric regions. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3 | 4    | 5   | 6    | 7 | 8 | 9 | 10 |
|---------|-----|---|------|-----|------|---|---|---|----|
| PEULER1 | PID |   | TYPE | SID | SID2 |   |   |   |    |

**Example:**

|         |     |  |       |     |  |  |  |  |  |
|---------|-----|--|-------|-----|--|--|--|--|--|
| PEULER1 | 100 |  | HYDRO | 300 |  |  |  |  |  |
|---------|-----|--|-------|-----|--|--|--|--|--|

**Descriptor      Meaning**

|      |  |
|------|--|
| PID  | Unique property number. (Integer > 0; Required)  |
| TYPE | They type of Eulerian material(s) being used. (Character; Default = HYDRO) <ul style="list-style-type: none"> <li>HYDRO      Hydrodynamic material + void.</li> <li>1STORDER    Single material, 1<sup>st</sup> order accurate Riemann solution-based fluids- &amp; gases solver.</li> <li>2NDORDER    Single material, 2<sup>nd</sup> order accurate Riemann solution-based fluids- &amp; gases solver.</li> <li>STRENGTH    Structural material with shear strength + void.</li> <li>MMHYDRO    Multimaterial hydrodynamic material with no shear strength + void.</li> <li>MMSTREN    Structural multimaterial with shear strength + void.</li> </ul> |
| SID  | Number of a TICEUL1 entry specifying the materials and geometric grouping criteria. (Integer > 0; Required)  |
| SID2 | Number of a EULFOR1 entry defining acceleration field. (Integer>=0 Default=0)  |

**Remarks:**

1. Remarks 1 through 6 of the PEULER definition apply also here.
2. Initial conditions and/or material assignments are defined on the TICEUL1 Bulk Data entry.

**PFAST**

## CFAST Fastener Property

Defines the CFAST fastener property values.

**Format:**

| 1     | 2   | 3   | 4    | 5     | 6     | 7    | 8     | 9   | 10 |
|-------|-----|-----|------|-------|-------|------|-------|-----|----|
| PFAST | PID | D   | MCID | MFLAG | KT1   | KT2  | KT3   | KR1 |    |
|       | KR2 | KR3 | MASS | GE    | ALPHA | TREF | COINL |     |    |

**Example:**

|       |   |     |    |  |         |        |        |  |  |
|-------|---|-----|----|--|---------|--------|--------|--|--|
| PFAST | 7 | 1.1 | 70 |  | 100000. | 46000. | 12300. |  |  |
|-------|---|-----|----|--|---------|--------|--------|--|--|

**Descriptor      Meaning**

|       |  |
|-------|--|
| PID   | Property identification number. (Integer > 0)  |
| D     | Diameter of the fastener. See Remark 2. (Real > 0)   |
| MCID  | Specifies the element stiffness coordinate system. See Remark 1. (Integer $\geq -1$ or blank; Default = -1)  |
| MFLAG | Defines if the coordinate system defined by MCID is absolute or relative. See Remark 1. (Integer 0 or 1; Default = 0)<br>If MFLAG = 0, MCID defines a relative coordinate system. See Remark 1a.<br>If MFLAG = 1, MCID defines an absolute coordinate system. See Remark 1c. |
| KTi   | Stiffness values in directions 1 through 3. (Real)   |
| KRi   | Rotational stiffness values in directions 1 through 3. (Real; Default = 0.0)   |
| MASS  | Lumped mass of fastener. (Real; Default = 0.0)   |
| GE    | Structural damping. (Real; Default = 0.0)  |
| ALPHA | Thermal expansion coefficient for the CFAST. (Real; Default=0.0)   |
| TREF  | Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 8.)   |
| COINL | Length of a CFAST with coincident grids. (Real; Default=0.0, COINL $\geq 0.0$ )  |

**Remarks:**

1.

- a. If MCID  $\geq 0$  and MFLAG = 0 (Default), then the KT1 stiffness will be applied along the  $x_{elem}$  axis direction of the fastener defined as

$$\hat{e}_1 = \frac{\hat{x}_B - \hat{x}_A}{\|\hat{x}_B - \hat{x}_A\|}$$

The T2 direction defined by MCID will be used to define the orientation vector  $\vec{v}$  of the fastener. Then the element  $z_{elem}$  axis will be defined as

$$\hat{\vec{e}}_3 = \frac{\hat{\vec{e}}_1 \times \vec{v}}{\|\hat{\vec{e}}_1 \times \vec{v}\|}$$

The KT3 stiffness will lie along the  $z_{elem}$  axis. The element  $y_{elem}$  axis is defined as

$$\hat{\vec{e}}_2 = \hat{\vec{e}}_3 \times \hat{\vec{e}}_1$$

The KT2 stiffness will lie along the  $y_{elem}$  axis

This option allows the user to define orthotropic material properties normal to the axis of the fastener that will “slide” with the curve of the patches.

- b. If MCID = -1, MFLAG is ignored, and the following element system is defined: the  $x_{elem}$  axis direction of the fastener defined as

$$\hat{\vec{e}}_1 = \frac{\vec{x}_B - \vec{x}_A}{\|\vec{x}_B - \vec{x}_A\|}$$

Relative to the basic system, find the smallest component  $j$  of the element  $x_{elem}$  axis unit vector. If two such components are equal, take the first one. Form a unit vector in the basic system. For example, assuming the  $j = 3$  component of  $\hat{\vec{e}}_1$  was the smallest.

$$b_j = b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

Form the following orthogonal vector:

$$\hat{\vec{e}}_2 = \vec{b}_j - \frac{\hat{\vec{e}}_1 \cdot \vec{b}_j}{\hat{\vec{e}}_1 \cdot \hat{\vec{e}}_1} \hat{\vec{e}}_1$$

$$\hat{\vec{e}}_2 = \frac{\hat{\vec{e}}_2}{\|\hat{\vec{e}}_2\|}$$

Form  $\hat{\vec{e}}_3$  as

$$\hat{\vec{e}}_3 = \hat{\vec{e}}_1 \times \hat{\vec{e}}_2$$

- c. If MCID  $\geq 0$  and MFLAG = 1, then the MCID will be used to compute stiffness. KT1 will be applied along the MCID T1 axis, KT2 along the MCID T2 axis, and KT3 along the MCID T3 axis. The element forces will be computed in the coordinate system defined in Remark 1b.
- d. If the length of GA - GB is zero, then the element  $x_{elem}$  axis is defined to lie along the projected normal to patch A.

2. The diameter D is used along with the piercing points of GA and GB to determine the location of fictitious grid points to form a fictitious hexa volume that determines the elements and physical grids used for the fastener element. Four points are positioned at  $\pm a$  positions parallel to the element axis where  $a = f(D)$ . The stiffness contribution of the fastener depends on both the stiffness values specified and the diameter D, because the  $\pm a$  positions are used along with the surface shape functions of the fictitious hexa to weight the contribution of the physical grids used to the grids GA and GB of the fastener element.
3. The CFAST element (see Figure 9-134), for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements. When  $\hat{x}_B - \hat{x}_A$  has finite length, internal rigid links connect grids GA and GB. This may result in coupling between translational and rotational degrees-of-freedom even when no rotational stiffness (KR1-KR3) are specified.  
For mass calculations, half the specified mass value is placed directly onto the projected grid A and grid B translational degrees-of-freedom.

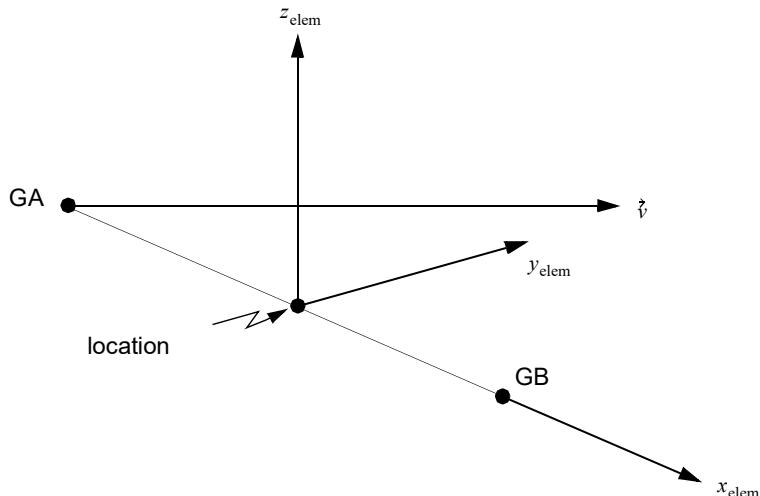


Figure 9-134 CFAST Element

4. The CFAST element lies midway between GA and GB.
5. Values for  $KT_i$  and  $KR_i$  are specified at the user's discretion. Assuming a short stubby beam where shear is dominate, possible values might be:

$$KT1 = \frac{EA}{L}$$

$$KT2 = \frac{G_2 A_s}{L}$$

$$KT3 = \frac{G_3 A_s}{L}$$

$$KR1 = \frac{GJ}{L}$$

$$KR2 = \frac{EI}{L} + \frac{G_2 A_s L}{3}$$

$$KR3 = \frac{EI}{L} + \frac{G_3 A_s L}{3}$$

where:

$$A = \pi D^2 / 4$$

$$I = \pi D^4 / 64$$

$$J = \pi D^4 / 32$$

$$L = |\vec{x}_B - \vec{x}_A|$$

$$A_s = A_s = A / \alpha_s$$

$$\alpha_s = 4 / 3$$

$E$ ,  $G_2$ ,  $G_3$ , and  $G$  are the material properties of the fastener.

The fastener stiffness is not, however, independent of the surrounding structure. The values of stiffness specified should not overwhelm the stiffness of the local structure or max ratio's will occur. One possible way to estimate the local stiffness  $S$  is by the relationship.

$$S = \frac{t_p E_p E}{E_p + E}$$

where  $t_p$  is a shell thickness and  $E_p$  is the modulus of the shell.

6. The element force and strain are computed as follows:

$$\{f_e\} = [K_e] \{u_e\} \text{ for statics}$$

$$\{f_e\} = ([K_e] + i(g + g_e)[K_e])(\{u_e\}_{\text{real}} + i\{u_e\}_{\text{imag}}) \text{ for frequency}$$

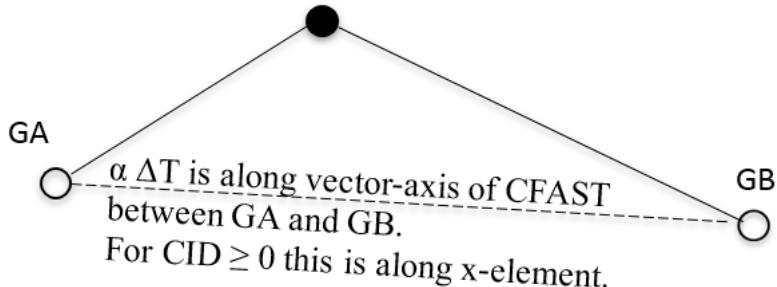
$$\{f_e\} = [K_e] \{u_e\} + \left( \frac{g}{w3} + \frac{g_e}{w4} \right) [K_e] \{v_e\} \text{ for transient}$$

where  $[K_e]$  is the  $6 \times 6$  element stiffness matrix,  $\{u_e\} = \{u_b\} - \{u_a\}$  relative displacement in the element coordinate system, and  $\{v_e\} = \{v_b\} - \{v_a\}$  relative velocity in the element coordinate system. The subscripts  $a$  and  $b$  stand for end A and end B of the fastener.  $g$  is defined by param,g;  $w3$  is defined by param,w3,  $w4$  is defined by param,w4; and  $g_e$  is the GE entry of the PFAST.  $\{u_e\}$  is the strain output. Stress output is the same as force output.

7. PFAST is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PFAST property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PFAST entries.

8. The thermal expansion for the CFAST will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERTURE Case Control command for thermal loading rules.

### Fast Location



**PFASTT**

## Frequency Dependent and material nonlinear CFAST Property

Defines the frequency dependent and material nonlinear properties for a PFAST Bulk Data entry.

**Format:**

| 1      | 2   | 3    | 4      | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|-----|------|--------|--------|--------|--------|--------|--------|----|
| PFASTT | PID | "K"  | TKID1  | TKID2  | TKID3  | TKID4  | TKID5  | TKID6  |    |
|        |     | "GE" | TGEID  |        |        |        |        |        |    |
|        |     | "KN" | TKNID1 | TKNID2 | TKNID3 | TKNID4 | TKNID5 | TKNID6 |    |
|        |     |      | FDC    | FUSE   | DIR    | OPTION | LOWER  | UPPER  |    |
|        |     |      | FSR    | LRGR   |        |        |        |        |    |

**Examples:**

|        |    |      |     |  |  |  |  |  |  |
|--------|----|------|-----|--|--|--|--|--|--|
| PFASTT | 33 | "GE" | 158 |  |  |  |  |  |  |
|--------|----|------|-----|--|--|--|--|--|--|

| Descriptor         | Meaning  |   |
|--------------------|--|---|
| PID                | Property identification number that matches the identification number on PFAST entry. (Integer > 0)  |   |
| "K"                | Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)  |   |
| TKID <sub>i</sub>  | Identification number of a TABLEDi entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)           |   |
| "GE"               | Flag indicating that the next field is a structural damping frequency table identification number. (Character)   |   |
| TGEID              | Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer ≥ 0; Default = 0) |   |
| "KN"               | Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)   |   |
| TKNID <sub>i</sub> | Identification number of a TABLEDi entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)              |   |
| FDC                | Force deflection curve rule. Specifies dependence between displacement components. See PBUSHT FDC entry of detail of rules.                                    |   |
| FUSE               | 0  | The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)                               |
|                    | 1  | The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for post processing.    |
|                    | 2  | The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from post processing. |

| Descriptor | Meaning   |
|------------|---|
| DIR        | The fuse direction. (0 < Integer < 6; Default = 0). 0 or 123456 imply any direction may fuse. Values of 1 through 6 may be placed in the field with no embedded blanks. See Remark 13. of PBUSHT.   |
| OPTION     | <p>Specifies a failure mode. (Character - Default="RELDIS")</p> <p>"ULTLD"      The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship.</p> <p>"RELDIS"      The specified max relative + displacement will be used to define failure.</p> <p>"RULTLD"      The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.</p> |
| LOWER      | <p>Lower failure bound. (Real; Default=0.0)</p> <p>If OPTION = "ULTLD" then LOWER specifies a lower failure load</p> <p>If OPTION = "RELDIS" then LOWER specifies a minimum relative displacement before failure.</p>   |
| UPPER      | <p>Upper failure bound. (Real; Default=0.0)</p> <p>If OPTION = "ULTLD" then UPPER specifies an upper failure load.</p> <p>If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.</p>   |
| FSR        | Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real > 0.0; Default = 1.-5)  |
| LRGR       | Specifies if large rotation is to occur at end A. (Integer > 0; Default = 0). Remark 14. of the PBUSHT.   |

### Remarks:

1. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PFAST entry.
2. The nominal values are used for all analysis types except frequency response. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. The PFAST entry may only be referenced by CFAST elements in the residual structure of SOL400 which do not attach to any omitted degrees-of-freedom.
4. For nonlinear analysis the nominal values for Ki on the PFAST should agree with the initial slope of the nonlinear force-displacement relationship defined by the PFAST, or the results will be unpredictable.

**PFTG**

## Fatigue Properties.

Defines fatigue properties of elements.

**Format:**

| 1    | 2     | 3      | 4      | 5       | 6  | 7      | 8      | 9 | 10 |
|------|-------|--------|--------|---------|----|--------|--------|---|----|
| PFTG | ID    |        | FINISH | KFINISH | KF | SCALE  | OFFSET |   |    |
|      | SHAPE | KTREAT | DIAM   | T1      | T2 | SPTFLG |        |   |    |

**Examples:**

|      |   |  |        |  |  |     |  |  |  |
|------|---|--|--------|--|--|-----|--|--|--|
| PFTG | 3 |  | POLISH |  |  | 1.2 |  |  |  |
|------|---|--|--------|--|--|-----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique ID referenced by a FTGDEF bulk data entry. (Integer>0)  |
| FINISH     | Material Surface Finish. This is a result of manufacturing process. Value can be NONE, POLISH, GROUND, MACHINE, POOR, ROLLED, CAST, KROUGH, KSURFC (Character; Default=NONE). See Remark 1.                                  |
| KFINISH    | Roughness factor for FINISH = KROUGH (0.0 < Real < 1.0, no Default). Surface roughness in microns for FINISH = KSURFC (Real > 0.0; no Default; 0.0 ≤ Real ≤ 1.0 for materials not listed in Remark 2.).                      |
| KF         | Fatigue strength reduction factor. (Real ≥ 0.0, Default=1.0). See Remark 3.  |
| SCALE      | Factor used to scale the resulting FE stresses of entities associated to this property set (Real; Default=1.0).  |
| OFFSET     | Offset used to offset the resulting FE stresses of entities associated to this property set (Real; Default=0.0).   |
| SHAPE      | Shape factor (Real ≥ 1.0; No default). See Remark 4.   |
| KTREAT     | Treatment factor (Real ≥ 0.0; Default = 1.0).  |
| DIAM       | Spot weld nugget diameter. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). See Remark 5. and 6.  |
| T1/T2      | Top (T1) and bottom (T2) thickness of shells connecting spot welds. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). Both should be left blank if either one is left blank. See Remark 5. |
| SPTFLG     | Flag to indicate that a lookup table is to be used to define the spot weld nugget diameter. 0 or 1, Default = 0, no lookup). Only used if CWELD elements are used to define spot welds. See Remark 6.                        |

**Remarks:**

1. The KF field can be used in lieu of or in addition to the FINISH & KTREAT field to modify the fatigue limit by multiplying the original fatigue limit by this value. POOR = Poor Machined. ROLLED = As Rolled. CAST = As Cast. KROUGH and KSURFC require that a KFINISH be entered. A material CODE on the MATFTG entry must be supplied to use anything other than NONE or POLISH, otherwise an error is issued.
2. If KFINISH = KSURFC, the user should enter a value for surface roughness  $R_z$  in  $\mu\text{m}$ . This is the average surface roughness according to the German standard DIN 4768. The Surface Roughness Factor  $K_r$  will then be calculated based on the strength and type of material (for example stronger materials are in general more sensitive to surface finish, and cast materials less so). The method for calculating  $K_r$  is taken from the FKM guideline *Analytical Strength Assessment of Components in Mechanical Engineering*.

If  $R_z \leq 1\mu\text{m}$ ,  $K_r = 1$ .

Otherwise:

$$K_r = 1 - a_r \log(R_z) \log(2R_m/R_{m,N,min})$$

$R_m$  is the UTS in MPa

$R_{m,N,min}$  and  $a_r$  are constants.

Table 9-30 Constants for Derivation of Surface Roughness Factor  $K_r$  from Roughness  $R_z$

| Material         | Steel       | GS*      | GGG† | GT‡  | GG** | Wrought Al Alloys | Cast Al alloys |
|------------------|-------------|----------|------|------|------|-------------------|----------------|
| CODE from MATFTG | 13,15,16-99 | 9-12, 15 | 5-8  | 2-4  | 1    | 100-105           | 106            |
| $a_r$            | 0.22        | 0.20     | 0.16 | 0.12 | 0.06 | 0.22              | 0.20           |
| $R_{m,N,min}$    | 400         | 400      | 400  | 350  | 100  | 133               | 133            |

\*GS = cast steel and heat treatable cast steel, for general purpose

†GG = cast iron with lamellar graphite (grey cast iron)

‡GGG = nodular cast iron

\*\*GT = malleable cast iron

3. Fatigue strength reduction factor can take into account notch effects, size effects, and loading type influence.  $K_f = C_{\text{notch}} \cdot C_{\text{size}} \cdot C_{\text{loading}}$  where the latter three are correction factors for each effect, respectively.
4. Setting the shape factor activates the Seeger Heuler plastic limit load correction. Leave this field blank if no plastic limit load correction is required.
5. If T1, T2, and DIAM are specifically supplied, they are used directly in the fatigue analysis of spot welds. If either T1 or T2 are blank, the thicknesses are automatically determined from the PSHELL entries that connect the spot welds.

When performing optimization using SOL 200, it is necessary to leave T1 and T2 blank if the connecting shell thicknesses are being used as design variable, otherwise the fatigue analysis of the spot welds will not use correct thicknesses from one design cycle to another.

6. If DIAM is left blank, the following rule is used:

- The diameter is derived based on the minimum thickness of the two sheets either side of the weld by performing a lookup on a table. This table is provided in the `spotweld.sys` file in the `msc20xx/<mach_arch>/nCode/nsyy` directory of a standard MSC Nastran installation. This is true for element types CBAR/CBEAM and CHEXA used to define spot welds. If using CWELD elements the diameter is automatically extracted from the PWELD entry, if SPTFLG=0. If SPTFLG=1, then the lookup table is used for CWELDs.

In the case of a table lookup to determine DIAM, the thicknesses of the top and bottom sheets must be within the range of the lookup table, which is up to 3mm by default. Otherwise a fatal error is issued. Also the diameter is set as a function of the thickness of the thinnest sheet joined by the spot weld. No interpolate between the data points in the `spotweld.sys` file is done; rather, the thickness of the thinnest sheet from each spot weld is compared to the table, and the value of thickness that is nearest to but less than or equal to the thickness of the sheet is identified. The corresponding diameter from the table is assigned to that spot weld.

Example `spotweld.sys` file:

```
SPOT WELD DEFINITION FILE
NUGGET DIAMETER=BY_THICKNESS
0.3,3.5
0.8,4.0
1.2,5.0
2.0,5.5
3.0,6.0
```

Note that there may be rounding errors in the extraction of sheet thicknesses, so when defining a `spotweld.sys` file, it may be a good idea to reduce the sheet thickness values by a small tolerance in order to avoid anomalous results

Table 9-31 Availability of Settings for Different Analysis Types

| Fatigue Property | Analysis / Material Type Category |    |          |          |
|------------------|-----------------------------------|----|----------|----------|
|                  | SN                                | EN | SpotWeld | SeamWeld |
| FINISH           | ✓                                 | ✓  | ✓        | ✓        |
| KFINISH          | ✓                                 | ✓† |          |          |
| KF               | ✓                                 | ✓† |          |          |
| SCALE            | ✓                                 | ✓  | ✓        | ✓        |
| OFFSET           | ✓                                 | ✓  | ✓        | ✓        |
| SHAPE            |                                   | ✓  |          |          |

Table 9-31 Availability of Settings for Different Analysis Types

| Fatigue Property | Analysis / Material Type Category |    |          |          |
|------------------|-----------------------------------|----|----------|----------|
|                  | SN                                | EN | SpotWeld | SeamWeld |
| KTREAT           | ✓                                 | ✓† |          |          |
| DIAM             |                                   |    | ✓        |          |
| T1/T2            |                                   |    | ✓        |          |
| SPTFLG           |                                   |    | ✓        |          |

✓ Available; † Not available for Multi-mean or Multi R-ratio curves

**PGAP****Gap Element Property**

Defines the properties of the gap element (CGAP entry).

**Format:**

| 1    | 2    | 3   | 4     | 5  | 6  | 7  | 8   | 9   | 10 |
|------|------|-----|-------|----|----|----|-----|-----|----|
| PGAP | PID  | U0  | F0    | KA | KB | KT | MU1 | MU2 |    |
|      | TMAX | MAR | TRMIN |    |    |    |     |     |    |

**Example:**

|      |   |      |     |      |  |      |      |      |  |
|------|---|------|-----|------|--|------|------|------|--|
| PGAP | 2 | .025 | 2.5 | 1.E6 |  | 1.E6 | 0.25 | 0.25 |  |
|------|---|------|-----|------|--|------|------|------|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| U0         | Initial gap opening. See <a href="#">Figure 9-136</a> . (Real; Default = 0.0)   |
| F0         | Preload. See <a href="#">Figure 9-136</a> . (Real $\geq$ 0.0; Default = 0.0)  |
| KA         | Axial stiffness for the closed gap; i.e., $U_a - U_b > U0$ . See <a href="#">Figure 9-136</a> . (Real > 0.0)  |
| KB         | Axial stiffness for the open gap; i.e., $U_a - U_b < U0$ . See <a href="#">Figure 9-136</a> . See Remark 2. (Real $\geq$ 0.0; Default = $10^{-14} \cdot KA$ )   |
| KT         | Transverse stiffness when the gap is closed. See <a href="#">Figure 9-137</a> . It is recommended that $KT \geq (0.1 \cdot KA)$ . (Real $\geq$ 0.0; Default = $MU1 \cdot KA$ )  |
| MU1        | Coefficient of static friction ( $\mu_s$ ) for the adaptive gap element or coefficient of friction in the y transverse direction ( $\mu_y$ ) for the nonadaptive gap element. See Remark 3. and <a href="#">Figure 9-137</a> . (Real $\geq$ 0.0; Default = 0.0)   |
| MU2        | Coefficient of kinetic friction ( $\mu_k$ ) for the adaptive gap element or coefficient of friction in the z transverse direction ( $\mu_z$ ) for the nonadaptive gap element. See Remark 3. and <a href="#">Figure 9-137</a> . (Real $\geq$ 0.0 for the adaptive gap element, $MU2 \leq MU1$ ; Default = $MU1$ ) |
| TMAX       | Maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment. See Remark 4. (Real; Default = 0.0)  |
| MAR        | Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See Remark 5. ( $1.0 < \text{Real} < 10^6$ ; Default = 100.0)   |
| TRMIN      | Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 6. ( $0.0 \leq \text{Real} \leq 1.0$ ; Default = 0.001)   |

**Remarks:**

1. [Figure 9-135](#), [Figure 9-136](#), and [Figure 9-137](#) show the gap element and the force-displacement curves used in the stiffness and force computations for the element.

2. For most contact problems, KA (penalty value) should be chosen to be three orders of magnitude higher than the stiffness of the neighboring grid points. A much larger KA value may slow convergence or cause divergence, while a much smaller KA value may result in inaccurate results. The value is adjusted as necessary if TMAX > 0.0.
3. When the gap is open, there is no transverse stiffness. When the gap is closed and there is friction, the gap has the elastic stiffness (KT) in the transverse direction until the friction force is exceeded and slippage starts to occur.
4. There are two kinds of gap elements: adaptive gap and nonadaptive gap. If TMAX  $\geq$  0.0, the adaptive gap element is selected by the program. When TMAX = 0.0, penalty values will not be adjusted, but other adaptive features will be active (i.e., the gap-induced stiffness update, gap-induced bisection, and subincremental process). The value of TMAX = -1.0 selects the nonadaptive (old) gap element. The recommended allowable penetration TMAX is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the gap.
5. The maximum adjustment ratio MAR is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K^{init}}{\text{MAR}} \leq K \leq K^{init} \cdot \text{MAR}$$

where  $K^{init}$  is either KA or KT.

6. TRMIN is used only for the penalty value adjustment in the adaptive gap element. The lower bound for the allowable penetration is computed by TRMIN · TMAX . The penalty values are decreased if the penetration is below the lower bound.

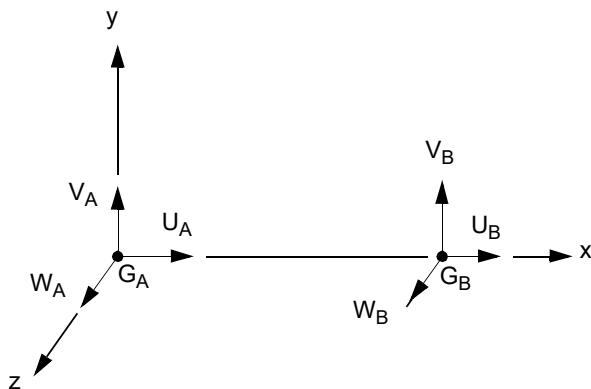


Figure 9-135 The CGAP Element Coordinate System

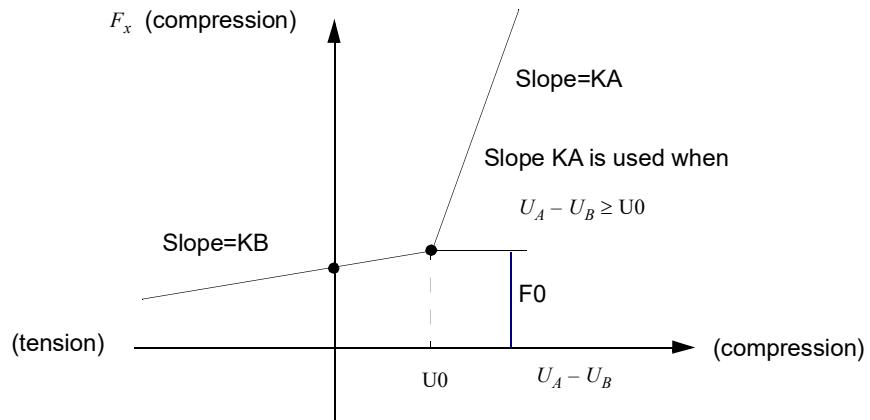


Figure 9-136 CGAP Element Force-Deflection Curve for Nonlinear Analysis

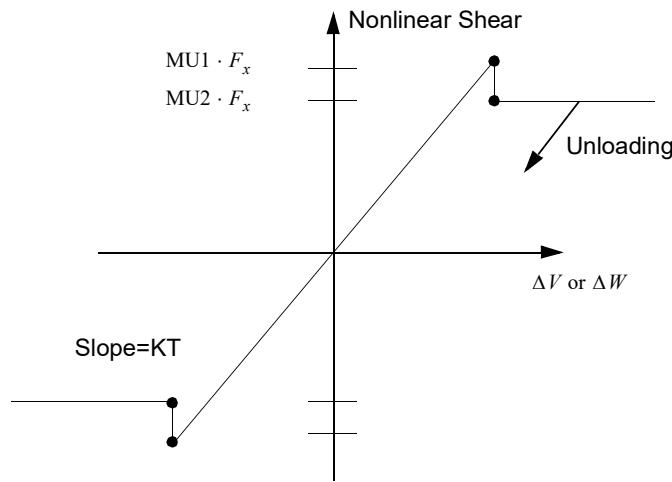


Figure 9-137 Shear Force for CGAP Element

7. If  $U_0$  is specified negative and GA and GB are not coincident, then the direction for closing must be controlled by the use of the CID field on the CGAP entry.
8. PGAP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PGAP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PGAP entries.

**PHBDY**

## CHBDYP Geometric Element Definition

A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----|----|----|---|---|---|---|----|
| PHBDY | PID | AF | D1 | D2 |   |   |   |   |    |

**Example:**

|       |   |     |     |     |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|
| PHBDY | 2 | .02 | 1.0 | 1.0 |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|

**Describer      Meaning**

|        |  |
|--------|--|
| PID    | Property identification number. (Unique Integer among all PHBDY entries).<br>(Integer > 0)   |
| AF     | Area factor of the surface used only for CHBDYP element TYPE = “POINT”,<br>TYPE = “LINE”, TYPE = “TUBE”, or TYPE = “ELCYL”. For TYPE = “TUBE”, AF is<br>the constant thickness of the hollow tube. (Real > 0.0 or blank) |
| D1, D2 | Diameters associated with the surface. Used with CHBDYP element TYPE = “ELCYL”,<br>“TUBE”, and “FTUBE”. (Real > 0.0 or blank; Default for D2 = D1)   |

**Remarks:**

1. The PHBDY entry is used with CHBDYP entries.
2. AF
  - For TYPE = “POINT” surfaces, AF is the area.
  - For TYPE = “LINE” or TYPE = “ELCYL” surfaces, AF is the effective width: area = AF · (length).
  - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces
$$\text{area} = \pi \cdot \left(\frac{D1 + D2}{2}\right) \cdot \sqrt{\left(LGTH\right)^2 + \left(\frac{D1 - D2}{2}\right)^2}$$
3. D1 and D2 are used only with TYPE = “ELCYL”, TYPE = “TUBE”, and TYPE = “FTUBE” surfaces.
  - For TYPE = “ELCYL” surfaces, D1 and D2 are the two diameters associated with the ellipse.
  - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces, D1 and D2 are the diameters associated with the first and second grid points, respectively.
  - For inner TYPE = “TUBE” surfaces, the diameters are reduced by twice the thickness ( $2 \cdot AF$ ) .

**PINTC**

## Properties of Geometric Interface -- Curve

Defines the properties for interface elements along curve interfaces between boundaries of multiple subdomains of p-elements.

**Format:**

| 1     | 2   | 3   | 4      | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|--------|---|---|---|---|---|----|
| PINTC | PID | TOL | DSCALE |   |   |   |   |   |    |

**Example:**

|       |   |      |        |  |  |  |  |  |  |
|-------|---|------|--------|--|--|--|--|--|--|
| PINTC | 1 | 0.01 | 1000.0 |  |  |  |  |  |  |
|-------|---|------|--------|--|--|--|--|--|--|

**Descriptor      Meaning**

- |        |  |
|--------|--|
| PID    | Property identification number. (Integer > 0)  |
| TOL    | Tolerance for distance between interface elements and subdomain boundaries. See Remark 2. (Real > 0.0; Default = 0.01) |
| DSCALE | Scaling parameter for Lagrange multiplier functions. See Remark 3. (Real > 0.0; Default = 1000.0)                      |

**Remarks:**

1. All PIDs must be unique.
2. TOL may be specified for the distance between the interface element and the boundaries. If the distance is greater than TOL, a warning message will occur. If the distance is less than TOL, but greater than the tolerance used by the geometric evaluator for the GMCURV method, a warning will be issued from the geometric evaluator.
3. DSCALE does not need to be specified unless the interface elements are poorly conditioned. Poor DSCALE conditioning can be determined from the epsilon value of the linear equation solution. A good value for DSCALE is two or three orders of magnitude less than the elastic moduli of the subdomain boundaries.

**PINTS****Properties of Geometric Interface -- Surface**

Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.

**Format:**

| 1     | 2   | 3   | 4      | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|--------|---|---|---|---|---|----|
| PINTS | PID | TOL | DSCALE |   |   |   |   |   |    |

**Example:**

|       |   |      |       |  |  |  |  |  |  |
|-------|---|------|-------|--|--|--|--|--|--|
| PINTS | 1 | 0.01 | 1000. |  |  |  |  |  |  |
|-------|---|------|-------|--|--|--|--|--|--|

| Descriptor | Meaning  | Type        | Default  |
|------------|--|-------------|----------|
| PID        | Property identification number   | Integer > 0 | Required |
| TOL        | Tolerance for distance between interface element and subdomain boundaries. | Real > 0    | 0.01     |
| DSCALE     | Scaling parameter for Lagrange multiplier functions.                       | Real > 0    | 1000.    |

**Remarks:**

1. All PIDs must be unique.
2. The TOL tolerance may be specified for the distance between the interface element and the subdomain boundaries. If the distance is greater than the TOL, a warning will be issued. If the distance is less than the TOL, but greater than the tolerance used by the geometric evaluator for the GMSURF, a warning from the geometric evaluator will be issued.
3. The DSCALE scaling parameter for the Lagrange multipliers does not need to be changed unless the interface elements are poorly conditioned. This could be determined from the epsilon value of the linear equation solution. A good value for DSCALE, which has the units of elastic modulus, is two or three orders of magnitude less than the elastic modulus of the subdomain boundaries.

**PLCOMP**

## Plane Strain or Axisymmetric Composite Element Property

Defines global (external) ply IDs and properties for a composite material laminate.

**Format:**

| 1      | 2    | 3      | 4       | 5      | 6     | 7 | 8 | 9 | 10 |
|--------|------|--------|---------|--------|-------|---|---|---|----|
| PLCOMP | PID  | DIRECT | THICKOP | SB     | ANAL  |   |   |   |    |
|        | "C4" | BEH4   | INT4    | BEH4H  | INT4H |   |   |   |    |
|        | "C8" | BEH8   | INT8    | BEH8H  | INT8H |   |   |   |    |
|        | ID1  | MID1   | T1      | THETA1 |       |   |   |   |    |
|        | ID2  | MID2   | T2      | THETA2 |       |   |   |   |    |

**Example:**

|        |      |     |    |      |  |  |  |  |  |
|--------|------|-----|----|------|--|--|--|--|--|
| PLCOMP | 782  | 1   |    |      |  |  |  |  |  |
|        | 1001 | 171 | .3 | 12.3 |  |  |  |  |  |
|        | 100  | 175 | .7 | 77.7 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| PID        | Element property identification number. (Integer > 0)   |
| DIRECT     | The layer direction for BEHi=COMPS or AXCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer input is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer $\pm 1$ or $\pm 2$ ; Default + 1) |
| THICKOP    | An out-of-plane thickness. (Real, Default = 1.0)  |
| ANAL       | Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL= "ISH" - Implicit structural and heat elements are being referred to. (Character Default ISH).   |
| SB         | Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real $\geq 0.0$ )  |
| C4         | Keyword indicating that two items following apply to elements with four corner grids. (Character)   |
| C8         | Keyword indicating that two items following apply to elements with four corner grids and four mid-side grids. (Character)   |
| BEHi       | Element structural behavior. See Remarks 4. and 7. (Character default: COMPS for BEH4 and BEH8)   |
| INTi       | Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H and INT8H)   |

| Descriptor | Meaning   |
|------------|---|
| BEHiH      | Element heat behavior. See Remark 8. (Character Default: COMPS for BEH4H and BEH8H)   |
| INTiH      | Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H, Q for INT8H)  |
| IDi        | Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)  |
| MIDi       | Material ID for the ply. See Remark 3. (Integer > 0)  |
| Ti         | Either fractional percent of the total element thickness or actual thickness of that ply depending on $\pm$ value of DIRECT. See Remarks 5. and 6. (Real > 0.0) |
| THETAi     | Orientation angle of the ply in the plane of the plies. These angles are measured about the thickness direction of the element. (Real; Default = 0.0)           |

**Remarks:**

1. The PLCOMP can only be referenced by a CQUAD, CQUAD4, CQUAD8, or CQUADX entry. Currently it should be used ONLY in SOL400. Erroneous results will occur in other Solution sequences.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across ALL PCOMPG, PLCOMP, and PCOMPLS entries.
3. The MIDi entry may point to MAT1, MAT3, MATHORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

| Implicit Structural Materials |         |         |       |        |         |
|-------------------------------|---------|---------|-------|--------|---------|
| MAT1                          | MAT3    | MATORT  | MATHE | MATUSR | MATDIGI |
| MATVE                         | <MATVE> | <MATVE> | MATVE |        |         |
| MATVP                         | MATVP   | MATVP   |       |        |         |
| MATEP                         | MATEP   | MATEP   |       |        |         |
| MATF                          | MATF    | MATF    |       |        |         |
| MATS1                         |         | MATSORT |       |        |         |

<MATVE> refers to the ALTERNATE format for type ORTHO

**Heat Materials**

MAT4 | MAT5

If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries may occur, between themselves, in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=COMPS or AXCOMP. A total of 1026 plies are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

| Layer Orientation |                      |   |                     |
|-------------------|----------------------|---|---------------------|
| DIRECT            | Normal to Layer edge | Layers run parallel from edge (ply numbering starts here) | to edge (ends here) |
| 1                 | Element Y direction  | G1-G2   | G4-G3               |
| 2                 | Element X direction  | G1-G4   | G2-G3               |

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and + 1. The ply thickness is entered in one of two ways:
  - a. Relative thickness where the numbers are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percentages of thickness must sum to 1.0.
  - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plies and then figures the fractional percent of each individual ply as in method 6a.
7. In the following table, BEHi refers to the structural behavior of 2D-solid elements. An underlined item delineates a default.

| Implicit Structural Classification of Elements |           |                  |              |         |
|--|-----------|------------------|--------------|---------|
| Element Structural Type                        | BEHi CODE | Integration Code | Element Type | # Nodes |
| Plane Strain composite                         | COMPS     | L                | QUAD         | 4       |
|  |           | Q                | QUAD         | 8       |
| Axisymmetric composite                         | AXCOMP    | L                | QUAD         | 4       |
|  |           | Q                | QUAD         | 8       |

8. In the following table, BEHiH refers to the heat behavior of 2D-solid elements. An underlined item delineates a default.

| Heat Classification of Elements |           |                  |              |         |
|---------------------------------|-----------|------------------|--------------|---------|
| Element Structural Type         | BEHi CODE | Integration Code | Element Type | # Nodes |
| Plane Strain composite          | COMPS     | <u>L</u>         | QUAD         | 4       |
|                                 |           | <u>Q</u>         | QUAD         | 8       |
| Axisymmetric composite          | AXCOMP    | L                | QUAD         | 4       |
|                                 |           | Q                | QUAD         | 8       |

9. Integration codes in Remark 7 are:

| INT CODE | Integration Type |
|----------|------------------|
| L        | Linear           |
| Q        | Quadratic        |

10. PLCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS, PLCOMP, PCOMPG, PSHELL, PLPLANE entries.

**PLOAD****Static Pressure Load**

Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

**Format:**

| 1     | 2   | 3 | 4  | 5  | 6  | 7  | 8 | 9 | 10 |
|-------|-----|---|----|----|----|----|---|---|----|
| PLOAD | SID | P | G1 | G2 | G3 | G4 |   |   |    |

**Example:**

|       |   |      |    |    |    |  |  |  |  |
|-------|---|------|----|----|----|--|--|--|--|
| PLOAD | 1 | -4.0 | 16 | 32 | 11 |  |  |  |  |
|-------|---|------|----|----|----|--|--|--|--|

**Descriptor      Meaning**

- SID              Load set identification number. (Integer > 0)  
 P              Pressure. (Real)  
 Gi              Grid point identification numbers. (Integer > 0; G4 may be zero or blank.)

**Remarks:**

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is blank, the surface is triangular.
3. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in [Figure 9-138](#).

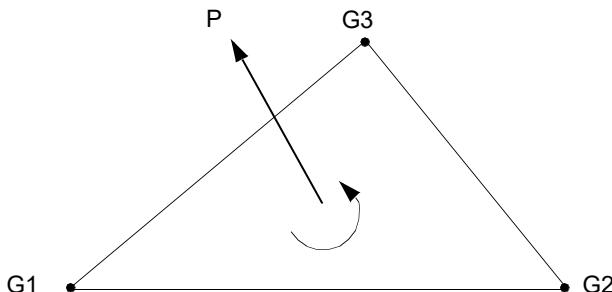
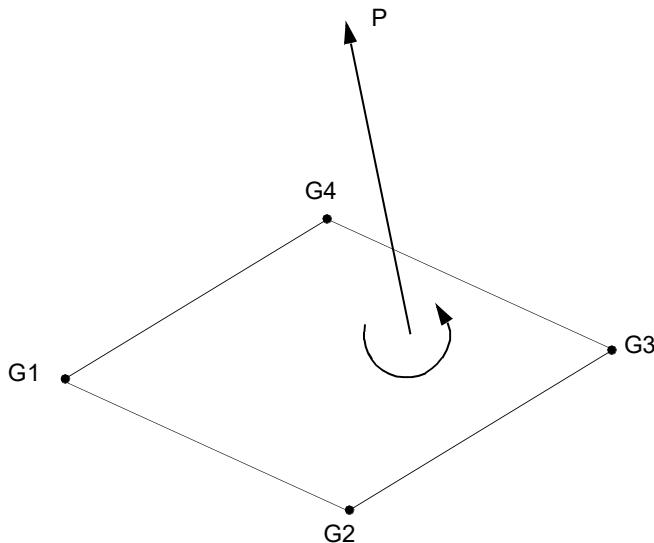


Figure 9-138 Pressure Convention for Triangular Surface of Surface Elements and/or the Faces of Solid Elements

The total load on the surface (see [Figure 9-139](#)), AP, is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

4. In the case of a quadrilateral surface, the grid points G1, G2, G3, and G4 should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:
  - The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.
  - One-half of the pressure is applied to each triangle, which is then treated in the manner described in Remark [2](#).



[Figure 9-139](#) Pressure Convention for Quadrilateral Surface of Surface Elements and/or the Faces of Solid Elements

5. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

**PLOAD1**

## Applied Load on CBAR, CBEAM or CBEND Elements

Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

**Format:**

| 1      | 2   | 3   | 4    | 5     | 6  | 7  | 8  | 9  | 10 |
|--------|-----|-----|------|-------|----|----|----|----|----|
| PLOAD1 | SID | EID | TYPE | SCALE | X1 | P1 | X2 | P2 |    |

**Example:**

|        |    |      |    |      |     |       |     |       |  |
|--------|----|------|----|------|-----|-------|-----|-------|--|
| PLOAD1 | 25 | 1065 | MY | FRPR | 0.2 | 2.5E3 | 0.8 | 3.5E3 |  |
|--------|----|------|----|------|-----|-------|-----|-------|--|

**Descriptor      Meaning**

|        |  |
|--------|--|
| SID    | Load set identification number. (Integer > 0)  |
| EID    | CBAR, CBEAM, or CBEND element identification number. (Integer > 0)   |
| TYPE   | Load type. (Character: "FX", "FY", "FZ", "FXE", "FYE", "FZE", "MX", "MY", "MZ", "MXE", "MYE", "MZE")             |
| SCALE  | Determines scale factor for X1, X2. (Character: "LE", "FR", "LEPR", "FRPR")                                      |
| X1, X2 | Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank; $0 \leq X1 \leq X2$ ) |
| P1, P2 | Load factors at positions X1, X2. (Real or blank)  |

**Remarks:**

1. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. If  $X2 \neq X1$ , a linearly varying distributed load will be applied to the element between positions X1 and X2, having an intensity per unit length of bar equal to P1 at X1 and equal to P2 at X2, except as noted in Remarks 8. and 9.
3. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.
4. If P1 = P2 and X2  $\neq$  X1, a uniform distributed load of intensity per unit length equal to P1 will be applied between positions X1 and X2 except as noted in Remarks 8. and 9.
5. Load TYPE is used as follows to define loads:  
"FX", "FY" or "FZ": Force in the x, y, or z direction of the basic coordinate system.  
"MX", "MY" or "MZ": Moment in the x, y, or z direction of the basic coordinate system.

- “FXE”, “FYE” or “FZE”: Force in the x, y, or z direction of the element’s coordinate system.
- “MXE”, “MYE” or “MZE”: Moment in the x, y, or z direction of the element’s coordinate system.
6. If SCALE = “LE” (length), the xi values are actual distances along the element axis, and, if  $X1 \neq X2$ , then Pi are load intensities per unit length of the element.
  7. If SCALE = “FR” (fractional), the xi values are ratios of the distance along the axis to the total length, and (if  $X2 \neq X1$ ) Pi are load intensities per unit length of the element.
  8. If SCALE = “LEPR” (length projected), the xi values are actual distances along the element axis, and (if  $X2 \neq X1$ ) the distributed load is input in terms of the projected length of the element.

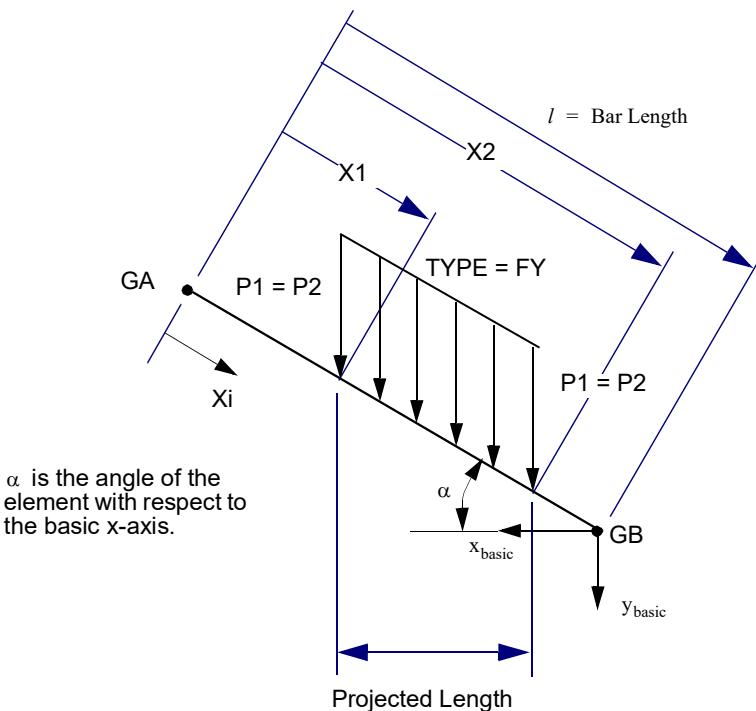


Figure 9-140 PLOAD1 Convention on Beam or Bar Elements

- If SCALE = “LE”, the total load applied to the bar is  $P_1 (X_2 - X_1)$  in the y-basic direction.
- If SCALE = “LEPR”, the total load applied to the bar is  $P_1 (X_2 - X_1) \cos\alpha$  in the y-basic direction.
9. If SCALE = “FRPR” (fractional projected), the Xi values are ratios of the actual distance to the length of the bar (CBAR entry), and if  $X1 \neq X2$ , then the distributed load is specified in terms of the projected length of the bar.
  10. Element identification numbers for CBAR, CBEAM, and CBEND entries must be unique.
  11. For the CBEND element, the following coordinate equivalences must be made for the element coordinates

$$R_{elem} \equiv X_{elem}$$

$$\theta_{elem} \equiv Y_{elem}$$

12. Only distributed loads applied over the entire length of the CBEND element may be applied.
13. Projected loads are not applicable to the CBEND element.
14. Loads on CBEAM elements defined with PLOAD1 entries are applied along the line of the shear centers.
15. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location  $X_i$  and output as a separate line. The force and stress locations  $X_i=0$  and  $X_i = l$  will always be output. This output format will be used for all beam and bar elements.
16. If on the TYPE field of the PLOAD1 entry, the element coordinate system direction (e.g. TYPE = FYE) option is selected, then the projection (i.e. SCALE = FRPR or LEPR) option is ignored and the result is the same as the SCALE = FR (or LE) option.

**PLOAD2**

## Uniform Normal Pressure Load on a Surface Element

Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.

**Format:**

| 1      | 2    | 3    | 4      | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|------|------|--------|------|------|------|------|------|----|
| PLOAD2 | SID  | P    | EID1   | EID2 | EID3 | EID4 | EID5 | EID6 |    |
|        | EID7 | EID8 | -etc.- |      |      |      |      |      |    |

**Example:**

|        |    |      |  |   |    |  |   |  |  |
|--------|----|------|--|---|----|--|---|--|--|
| PLOAD2 | 21 | -3.6 |  | 4 | 16 |  | 2 |  |  |
|--------|----|------|--|---|----|--|---|--|--|

**Alternate Format and Example:**

|        |     |      |      |        |      |  |  |  |  |
|--------|-----|------|------|--------|------|--|--|--|--|
| PLOAD2 | SID | P    | EID1 | “THRU” | EID2 |  |  |  |  |
| PLOAD2 | 1   | 30.4 | 16   | THRU   | 48   |  |  |  |  |

**Descriptor      Meaning**

- SID      Load set identification number. (Integer > 0)  
 P      Pressure value. (Real)  
 EIDi      Element identification number. (Integer  $\geq 0$  or blank; for the “THRU” option, EID1 < EID2.)

**Remarks:**

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. At least one positive EID must be present on each PLOAD2 entry.
3. If the alternate form is used, all elements EID1 through EID2 must be two-dimensional.
4. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry. Refer to the PLOAD entry.
5. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
6. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution

sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

7. The PLOAD2 entry may not be applied to p-elements. The PLOAD4 must be used.

**PLOAD4****Pressure Load on Surface and Faces of Solid Elements**

Defines a pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element. See Remark 17.

**Format:**

| 1      | 2   | 3   | 4  | 5  | 6    | 7    | 8  | 9        | 10 |
|--------|-----|-----|----|----|------|------|----|----------|----|
| PLOAD4 | SID | EID | P1 | P2 | P3   | P4   | G1 | G3 or G4 |    |
|        | CID | N1  | N2 | N3 | SORL | LDIR |    |          |    |

**Example:**

|        |   |      |      |     |     |  |    |  |  |
|--------|---|------|------|-----|-----|--|----|--|--|
| PLOAD4 | 2 | 1106 | 10.0 | 8.0 | 5.0 |  | 48 |  |  |
|        | 6 | 0.0  | 1.0  | 0.0 |     |  |    |  |  |

**Alternate Format and Example (See Remark 8.):**

|        |     |      |    |    |      |      |        |      |  |
|--------|-----|------|----|----|------|------|--------|------|--|
| PLOAD4 | SID | EID1 | P1 | P2 | P3   | P4   | "THRU" | EID2 |  |
|        | CID | N1   | N2 | N3 | SORL | LDIR |        |      |  |

|        |   |      |      |     |     |  |      |      |  |
|--------|---|------|------|-----|-----|--|------|------|--|
| PLOAD4 | 2 | 1106 | 10.0 | 8.0 | 5.0 |  | THRU | 1143 |  |
|        | 6 | 0.0  | 1.0  | 0.0 |     |  |      |      |  |

| Descriptor                    | Meaning   |
|-------------------------------|---|
| SID                           | Load set identification number. (Integer > 0)   |
| EID<br>EID1<br>EID2 }<br>EID2 | Element identification number. (Integer > 0; for the "THRU" option, EID1 < EID2)  |
| P1, P2, P3, P4                | Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1.)   |
| G1                            | Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)  |
| G3                            | Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA, CPENTA or CPYRAM element. Required data for quadrilateral faces of CHEXA, CPENTA and CPYRAM elements only. For CPYRAM element triangle faces, G1 and G3 are adjacent corner nodes on the quadrilateral face, and the load is applied on the triangular face which includes those grids. G3 must be omitted for a triangular surface on a CPENTA element. |
| G4                            | Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)  |

| Descriptor | Meaning  |
|------------|--|
| CID        | Coordinate system identification number. See Remark 2. (Integer $\geq 0$ ; Default = 0)  |
| N1, N2, N3 | Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark 2. (Real)  |
| SORL       | The character string SURF or LINE. SURF means the surface load acting on the surface of the element and LINE means the consistent edge loads acting on the edges of the element. The default is SURF. See Remark 13. |
| LDIR       | Denote the direction of the line load (SORL=LINE), character string X, Y, Z, TANG, or NORM. The default is NORM. See Remark 14.  |

### Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
4. For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)
5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
6. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
7. G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.
8. The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.

9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.
10. For the CQUADR and CTRIAR element, only pressure that acts normal to the element is computed properly. Surface tractions are not resolved into moments normal to the element.
11. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
12. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 829](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
13. The SORL field is ignored by all elements except QUADR and TRIAR. For QUADR or TRIAR only, if SORL=LINE, the consistent edge loads are defined by the PLOAD4 entry. P1, P2, P3 and P4 are load per unit length at the corner of the element. If all four Ps are given, then the line loads along all four edges of the element are defined. If any P is blank, then the line loads for only two edges are defined. For example, if P1 is blank, the line loads of the two edges connecting to G1 are zero. If two Ps are given, then the line load of the edge connecting to the two grid points is defined. If only one P is given, the second P value default to the first P value. For example, P1 denotes that the line load along edge G1 and G2 has the constant value of P1.
14. The direction of the line load (SORL=LINE) is defined by either (CID, N1, N2, N3) or LDIR. Fatal error will be issued if both methods are given. TANG denotes that the line load is in tangential direction of the edge, pointing from G1 to G2 if the edge is connecting G1 and G2. NORM denotes that the line load is in the mean plan, normal to the edge, and pointing outward from the element. X, Y, or Z denotes the line load is in the X, Y, or Z direction of the element coordinate system. If both (CID, N1, n2, N3) and LDIR are blank, then the default is LDIR=NORM.
15. For SOL 600, the SORL field may also be used by CQUAD4 and CTRIA3 in addition to CQUADR and CTRIAR.
16. For SOL 600, the LDIR field must be blank or a fatal error will occur. SOL 600 line loads must use the CID, N1, N2, N3 fields.
17. For SOL 400, with large displacement (PARAM,LGDISP), higher-order 6-node triangles (CTRIA6), 8-node quadrilateral (CQUAD8), 20-node hexagonal (CHEXA), or 15-node pentahedral (CPENTA), the load is not calculated correctly. Use lower-order elements if distributed loads are required in a large displacement analysis.

**PLOADB3**

Applied distributed load on CBEAM3 elements

Defines a distributed load to a CBEAM3 element over entire length of the beam axis.

**Format:**

| 1       | 2    | 3    | 4    | 5  | 6  | 7  | 8    | 9     | 10 |
|---------|------|------|------|----|----|----|------|-------|----|
| PLOADB3 | SID  | EID  | CID  | N1 | N2 | N3 | TYPE | SCALE |    |
|         | P(A) | P(B) | P(C) |    |    |    |      |       |    |

**Example:**

|         |      |      |       |     |  |  |        |  |  |
|---------|------|------|-------|-----|--|--|--------|--|--|
| PLOADB3 | 10   | 1002 | LOCAL | 1.0 |  |  | MOMENT |  |  |
|         | 100. | 90.  | 70.   |     |  |  |        |  |  |

**Descriptor      Meaning**

|            |  |
|------------|--|
| SID        | Load set identification number. (Integer > 0; Required)  |
| EID        | CBEAM3 element identification number. (Integer > 0, Required)  |
| CID        | Coordinate system for load definition. (Character or Integer; Default = "BASIC")<br>"LOCAL"            Local coordinate system;<br>"ELEMENT"        Element coordinate system;<br>"BASIC" or 0      Basic coordinate system; |
| n (n>0):   | Any user-specified coordinate system identification number.  |
| N1, N2, N3 | Load vector components measured in coordinate system specified by CID. (Real; at least one Ni ≠ 0.0)   |
| TYPE       | Type of applied load. (Character = "FORCE", "MOMENT" or "BIMOMENT"; Required)  |
| SCALE      | Load vector scale factor. (Real; Default = 1.0)  |
| P(j)       | Magnitudes of load at j (j=A, B and C). (Real; Default = 0.0)  |

**Remarks:**

1. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. The load vector is defined by  $\vec{P}_j = SCALE \cdot P_j \cdot \vec{N}$ , (j = A, B, C). The orientation of load  $\vec{P}$  is determined by vector  $\vec{N}$  and the magnitude is equal to  $SCALE \cdot P$  times magnitude of vector  $\vec{N}$ .

3. The distributed load is applied over the entire length of the beam axis, along the line of the shear center.

**PLOADX1**

## Pressure Load on Axisymmetric Element

Defines surface traction to be used with CAXISYM, CQUADX, CTRIAZ, and CTRIAZ6 axisymmetric elements.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6  | 7  | 8     | 9 | 10 |
|---------|-----|-----|----|----|----|----|-------|---|----|
| PLOADX1 | SID | EID | PA | PB | GA | GB | THETA |   |    |

**Example:**

|         |     |    |     |      |    |    |     |  |  |
|---------|-----|----|-----|------|----|----|-----|--|--|
| PLOADX1 | 200 | 35 | 3.5 | 10.5 | 10 | 30 | 20. |  |  |
|---------|-----|----|-----|------|----|----|-----|--|--|

**Descriptor      Meaning**

|        |   |
|--------|---|
| SID    | Load set identification number. (Integer > 0)   |
| EID    | Element identification number. (Integer > 0)  |
| PA     | Surface traction at grid point GA. (Real)   |
| PB     | Surface traction at grid point GB. (Real; Default = PA)   |
| GA, GB | Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0) |
| THETA  | Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0)         |

**Remarks:**

1. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. PLOADX1 is intended only for the CAXISYM, CQUADX, CTRIAZ, and CTRIAZ6 elements.
3. The surface traction is assumed to vary linearly along the element side between GA and GB.
4. The surface traction is input as force per unit area. MSC Nastran converts this traction into equivalent nodal loads by integration over the surface. For AXISYM and hyperelastic QUADX and TRIAX elements this integration is over one radian. For TRIAX6 and axisymmetric harmonic QUADX and TRIAX elements this integration is over two pi radians.
5. THETA is measured counter-clockwise from the inward normal of the straight line between GA and GB, to the vector of the applied load, as shown in [Figure 9-141](#), [Figure 9-142](#), and [Figure 9-143](#). Positive pressure is in the direction of inward normal to the line segment.

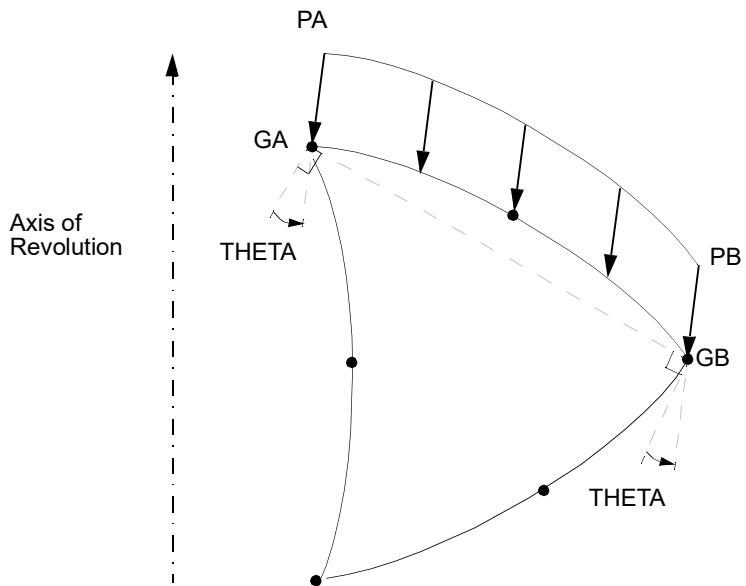


Figure 9-141 Pressure Load on CTRIA6 or CTRIAX Element

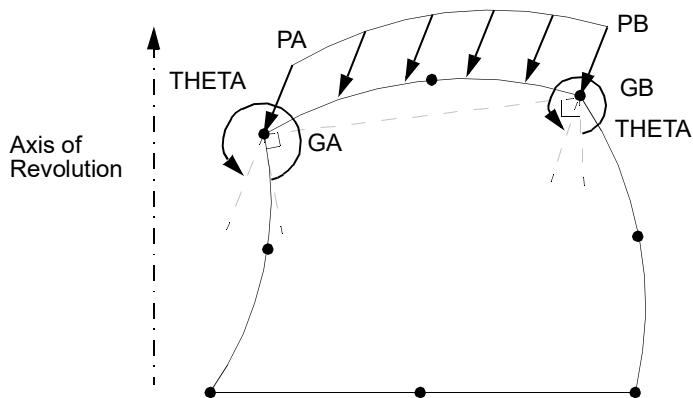


Figure 9-142 Pressure Load on CQUADX Element

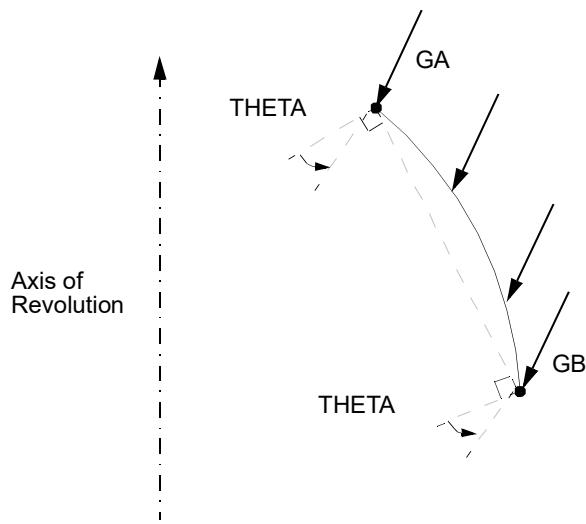


Figure 9-143 Pressure Load on CAXISYM Element

6. Axisymmetric harmonic elements may have Harmonic "N" greater than zero. For this case the "PA" and "PB" fields of the PLOADX1 entry are coefficients of  $\cos(N*\theta)$ . The PLOADX1 cannot supply azimuthal tractions on axisymmetric harmonic elements.

**PLOTEL**

## Dummy Plot Element Definition

Defines a one-dimensional dummy element for use in plotting.

**Format:**

| 1      | 2   | 3  | 4  | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|----|----|---|---|---|---|---|----|
| PLOTEL | EID | G1 | G2 |   |   |   |   |   |    |

**Example:**

|        |    |    |    |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|
| PLOTEL | 29 | 35 | 16 |  |  |  |  |  |  |
|--------|----|----|----|--|--|--|--|--|--|

**Descriptor      Meaning**

- EID            Element identification number. (Integer > 0)  
G1, G2        Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

**Remarks:**

1. This element is not used in the model during any of the solution phases of a problem. It is used to simplify plotting of structures with large numbers of colinear grid points, where the plotting of each grid point along with the elements connecting them would result in a confusing plot.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one PLOTEL element may be defined on a single entry.
4. In superelement analysis, PLOTELs, as well as other elements such as CBAR, CQUAD4, etc., will affect the formation of the superelement tree. The PLOTEL EIDs will also appear in the superelement map output; see the description of PARAM,SEMAPRPT in [Parameters, 783](#).
5. Only grid points connected by structural elements appear on structure plots. This does not include points connected only by rigid or general elements or MPCs. A plot element in parallel with elements that do not plot will cause these points to be present.

**PLPLANE**

## Fully Nonlinear Plane Element Properties

Defines the properties of a fully nonlinear (i.e., large strain and large rotation, etc.) plane strain, plane stress, or axisymmetric element. Please refer to PSHLN2.

**Format:**

| 1       | 2   | 3   | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-----|-----|-----|---|---|---|---|----|
| PLPLANE | PID | MID | CID | STR |   |   |   |   |    |

**Example:**

|         |     |     |    |      |  |  |  |  |  |
|---------|-----|-----|----|------|--|--|--|--|--|
| PLPLANE | 203 | 204 | -2 | GRID |  |  |  |  |  |
|---------|-----|-----|----|------|--|--|--|--|--|

**Descriptor      Meaning**

|     |  |
|-----|--|
| PID | Element property identification number. (Integer > 0)  |
| MID | Identification number of a MATHP entry for Nastran conventional elements, or MATHE and other linear or nonlinear material entries for Nastran advanced nonlinear elements. (Integer > 0). See Remarks 7. |
| CID | Identification number of a coordinate system defining the plane of deformation. See Remarks 2. and 3. (Default = 0)  |
| STR | Location of stress and strain output. (Character: "GAUS" or "GRID", Default = "GRID")  |

**Remarks:**

1. PLPLANE can be referenced by a CQUAD, CQUAD4, CQUAD8, CQUADX, CTRIA3, CTRIA6, or CTRIAX entry.
2. Plane strain hyperelastic elements must lie on the x-y plane of the CID coordinate system. Stresses and strains are output in the CID coordinate system.
3. Axisymmetric hyperelastic elements must lie on the x-y plane of the basic coordinate system. CID may not be specified and stresses and strains are output in the basic coordinate system.
4. Negative values of CID will indicate an axes combination for axisymmetric elements. The following table describes the value and axes. Inside one model, only one CID value is allowed.

| Axial | Radial | CID Value |
|-------|--------|-----------|
| X     | Y      | -2        |
| X     | Z      | -3        |
| Y     | X      | -4        |

| Axial | Radial | CID Value |
|-------|--------|-----------|
| Y     | Z      | -5        |
| Z     | X      | -6        |
| Z     | Y      | -7        |

5. CID = 0 is the default value which equal to CID = -2.
6. PLPLANE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLPLANE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness among PLPLANE, PAXSYMH, and PSHELL entries.
7. When using Enhanced materials (SOL400), the MID field may point to additional Material entries such as a MAT1. For a full list see the PSHLN2 entry. If a material type other than the MATHP or MATHE entry is used in solution sequences other than SOL400, wrong results will occur. Note linear stress recovery is not available for any element using the PLPLANE.

## PLSOLID

## Fully Nonlinear Solid Element Properties

Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.

### Format:

| 1       | 2   | 3   | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-----|-----|---|---|---|---|---|----|
| PLSOLID | PID | MID | STR |   |   |   |   |   |    |

### Example:

|         |    |    |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|
| PLSOLID | 20 | 21 |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|

### Descriptor      Meaning

- |     |   |
|-----|---|
| PID | Element property identification number. (Integer > 0)                                 |
| MID | Identification number of a MATHP entry. (Integer > 0)                                 |
| STR | Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”) |

### Remarks:

1. PLSOLID can be referenced by a CHEXA, CPENTA, or CTETRA entry.
2. Stress and strain are output in the basic coordinate system.
3. PLSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PLSOLID entries.

**PLTSURF****Surface plot mesh definition**

Defines the surface plot mesh.

**Format:**

| 1       | 2    | 3    | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|---------|------|------|------|------|------|------|------|------|----|
| PLTSURF | EID  | GID1 | GID2 | GID3 | GID4 | GID5 | GID6 | GID7 |    |
|         | GID8 |      |      |      |      |      |      |      |    |

**Descriptor      Meaning**

EID              Identification number of element.

GID<sub>i</sub>              Grid identification number of structure/cavity models or trim component surface element.

**Remarks:**

1. This element is used only to describe the surface mesh of 3D model. The connection can be 3, 4, 6 or 8 nodes and the connection sequence mirrors those of CTRIA3, CQUAD4, CTRIA6 and CQUAD8.
2. PLTSURF can be utilized to describe surface mesh on ACPEMCP for trim component coupling in PEM job.
3. For trim component, PLTSURF ID must be referenced on SET3,id,ELEM. If SET1 entry is used with PLTSURF ID, fatal error or unexpected results may be the results.
4. PLTSURF entry can also be used to define surface mesh for structure/cavity models and it behaves similarly as PLOTEL and does not participate in analysis.

**PMARKER**

## Property Definition of a Marker Element

Defines the behavior of the marker element in the EULER domain. Used in SOL 700 only.

**Format:**

| 1       | 2  | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|------|---|---|---|---|---|---|----|
| PMARKER | ID | TYPE |   |   |   |   |   |   |    |

**Example:**

|         |   |       |  |  |  |  |  |  |  |
|---------|---|-------|--|--|--|--|--|--|--|
| PMARKER | 7 | FIXED |  |  |  |  |  |  |  |
|---------|---|-------|--|--|--|--|--|--|--|

**Describer      Meaning**

- |        |  |
|--------|--|
| ID     | Marker property ID, referred by CMARKB2 and CMARKN1 entries. (Integer > 0, Required) |
| TYPE   | Behavior of the marker grid points in the Euler domain: (Character, Default = FIXED) |
| FIXED  | The marker will not move in the Euler domain.  |
| MOVING | The marker will be moved by velocities in the Euler domain.                          |

**Remarks:**

1. The PMARKER entry will be ignored for elements that are referring to structural grid points. These structural grid points will move with the structure and the Euler velocities do not change their velocity.
2. TYPE = FIXED. This means that the marker is stationary through out the simulation and it is therefore not moving with the Euler velocity. If the marker grid is located outside the Eulerian domain(s), the Marker will still be allowed to exist. It means, however, that no variables are recorded and that the variables will appear as zero on the Time History plots.
3. TYPE = MOVING. The marker is moving along with the Eulerian material. When the grid point approaches a coupling surface there is no mechanism that prevents the marker from passing through the coupling surface. When this happens the marker enters an element that is covered and motion of the grid point will stop. It is allowed that the grid point moves from one Euler domain to the other through a porous hole or a coupling surface with interactive failure.

**PMASS**

## Scalar Mass Property

Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

**Format:**

| 1     | 2    | 3  | 4    | 5  | 6    | 7  | 8    | 9  | 10 |
|-------|------|----|------|----|------|----|------|----|----|
| PMASS | PID1 | M1 | PID2 | M2 | PID3 | M3 | PID4 | M4 |    |

**Example:**

|       |   |      |   |      |  |  |  |  |  |
|-------|---|------|---|------|--|--|--|--|--|
| PMASS | 7 | 4.29 | 6 | 13.2 |  |  |  |  |  |
|-------|---|------|---|------|--|--|--|--|--|

| Descriptor | Meaning |
|------------|---------|
|------------|---------|

|                  |   |
|------------------|---|
| PID <sub>i</sub> | Property identification number. (Integer > 0) |
|------------------|---|

|                |                              |
|----------------|------------------------------|
| M <sub>i</sub> | Value of scalar mass. (Real) |
|----------------|------------------------------|

**Remarks:**

1. Mass values are defined directly on the CMASS2 and CMASS4 entries, and therefore do not require a PMASS entry.
2. Up to four mass values may be defined by this entry.
3. For a discussion of scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
4. PMASS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PMASS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PMASS entries.

**PMINC**

## Constant Spallation Model

Defines a spallation model where the minimum pressure is constant. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3     | 4     | 5      | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-------|-------|--------|---|---|---|---|----|
| PMINC | PID | VALUE | FVTOL | FVTOL2 |   |   |   |   |    |

**Example:**

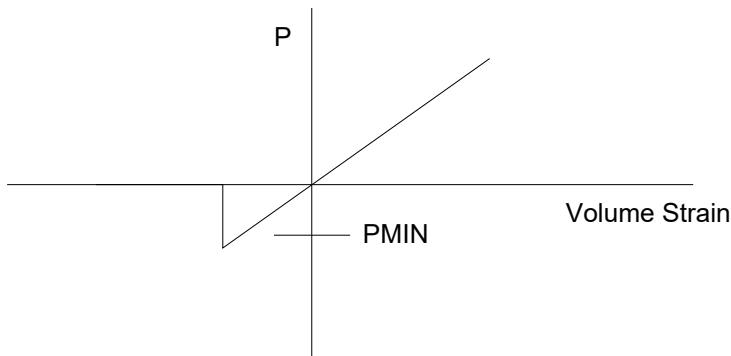
|       |     |      |  |  |  |  |  |  |  |
|-------|-----|------|--|--|--|--|--|--|--|
| PMINC | 220 | -370 |  |  |  |  |  |  |  |
|-------|-----|------|--|--|--|--|--|--|--|

**Descriptor      Meaning**

|        |  |
|--------|--|
| PID    | Unique PMINC number. Referenced from MATDEUL. (Integer > 0, Required)                    |
| VALUE  | The value of the minimum pressure. (Real $\leq 0.0$ , Default = 0.0)                     |
| FVTOL  | Volume cutoff tolerance. (Real $> 0$ , 1.E-4)  |
| FVTOL2 | Maximum void fraction that is permissible under tension. (Real $\geq 0$ ; Default = 0.0) |

**Remarks:**

1. If the pressure in an element falls below the minimum pressure, the element spalls. The pressure and yield stress are set to zero.



2. The default for the volume cutoff tolerance is 1.E-4. This value should be decreased in case of large mass increase of material without any reason.
3. If an element spalls a void is created. In order to prevent getting void fractions that are too small, a void fraction is put to zero if it is smaller than FVTOL. The default for FVTOL is 1.E-4 and works only for Eulerian elements. This value should be decreased in case of large mass increase of material without any reason. Voids can be created during transport of material, because of a material failure and by unloading.

4. With FVTOL2 = 0, any void fraction in an element will lead to failure, and then no tensile stresses are possible. In simulations in which tensile conditions are present, it can be required to allow for tensile stresses in the presence of a small void fraction not exceeding a threshold. This threshold is given by FVTOL2. A good value for FVTOL2 = 2 \* FVTOL = e.E-4. FVTOL 2 is only used for Eulerian materials. FVTOL 2 should be larger than FVTOL.

**PMREBAI**

## Defines Rebar Property Information for CMREBAI Elements in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. SOL 600 only.

**Format:**

| 1       | 2      | 3    | 4     | 5    | 6    | 7      | 8     | 9      | 10 |
|---------|--------|------|-------|------|------|--------|-------|--------|----|
| PMREBAI | IP     | TOL  | IFILE | X1   | X2   | X3     | MICRO | FACTOR |    |
|         | IM1    | POS1 | AREA1 | SSS1 | ANG1 | IPOST1 | IORI1 |        |    |
|         | IM2    | POS2 | AREA2 | SSS2 | ANG2 | IPOST2 | IORI2 |        |    |
|         | IM3    | POS3 | AREA3 | SSS3 | ANG3 | IPOST3 | IORI3 |        |    |
|         | IM4    | POS4 | AREA4 | SSS4 | ANG4 | IPOST4 | IORI4 |        |    |
|         | IM5    | POS5 | AREA5 | SSS5 | ANG5 | IPOST5 | IORI5 |        |    |
|         | "HEXA" | ID1  | THRU  | ID2  | IHE3 | THRU   | IHE4  |        |    |
|         | "HEXA" | ID5  | THRU  | ID6  | etc. |        |       |        |    |

**Example (two rebar layers through matrix CHEXA elements 100-120):**

|         |      |       |       |      |         |      |   |      |  |
|---------|------|-------|-------|------|---------|------|---|------|--|
| PMREBAI | 20   |       | 1     | 1.0  | 0.0     | 0.0  | 1 | 0.01 |  |
|         | 100  | 0.25  | 0.075 | 3.0  | 45.0    | 2000 | 1 |      |  |
|         | 101  | 0.275 | 0.050 | 32.0 | 4 -30.0 | 2001 | 1 |      |  |
|         | HEXA | 100   | THRU  | 120  |         |      |   |      |  |

**Descriptor      Meaning**

- IP                  Rebar property ID. (Integer, Required, no Default)
- TOL                Exterior tolerance. A rebar grid is considered within a matrix CHEXA element if the distance between the element and the grid is smaller than the tolerance times the average edge length of the element unless the grid is actually inside another matrix CHEXA element. (Real, Default = 0.05)
- IFILE             Option to create a file for rebar verification. (Integer, Default = 0). If IFILE=1, a file named jid.marc\_rebar.mfd will be created for use by Mentat.
- X1                First direction cosine of reference axis. (Real, Required, no Default)
- X2                Second direction cosine of reference axis. (Real, Required, no Default)
- X3                Third direction cosine of reference axis. (Real, Required, no Default)
- MICRO            Option to activate "micro buckling" behavior. (Integer, Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
- FACTOR           Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real, Default = 0.02)

|                |  |
|----------------|--|
| IMi            | Material identification number. (Integer, Required, no Default)  |
| POSi           | Relative position of rebar layer at edge 1 (pr/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real, Required, no Default)   |
| AREAi          | Rebar cross sectional area. (Real, Required, no Default)   |
| SSSi           | Number of rebars per unit length in each layer. (Real, Required, no Default)   |
| ANGi           | Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real, Default = 0.0, must be between -90.0 and 90.0), see <a href="#">Figure 9-144</a> .  |
| IPOSTi         | Global identification number of the rebar layer used for postprocessing. (Integer, Default = 0)  |
| IORIi          | Rebar layer orientation type, see <a href="#">Figure 9-145</a> . (Integer, Required, no Default)<br>1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face<br>2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face<br>3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face |
| HEXA           | Enter the string HEXA to indicate the line that contains matrix CHEXA element ID's through which the rebar passes. (Character, Required)   |
| ID1, ID3, etc. | Starting matrix CHEXA element ID. (Integer ID1 is Required)  |
| THRU           | Enter string THRU to indicate that more than one matrix CHEXA element with these rebar properties apply.   |
| ID2, ID4, etc. | Ending matrix CHEXA element ID. (Integer, ID2 is Required)   |

### Remarks:

1. This entry is the property entry for CMREBAI elements and makes use of Marc's REBAR and INSERT capabilities for rebar membrane element types 147 and 148. Entries CMREBAR and PMREBAR makes use of Marc's REBAR only capability and use rebar elements 23 and 146.
2. If rebar layers i are not desired, the lines 2-5 may be omitted. The line for i=1 is required.
3. See the following two figures for a definition of ANG and IORI.
4. Any material valid for SOL 600 may be used to define IM.
5. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords).  $\alpha$  is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the next figure.

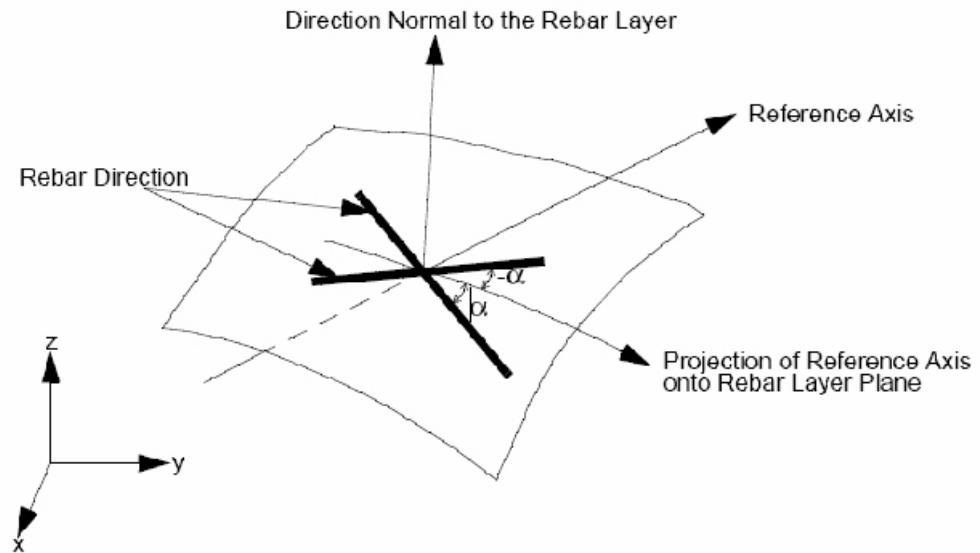


Figure 9-144 Description of rebar orientation on a single rebar layer

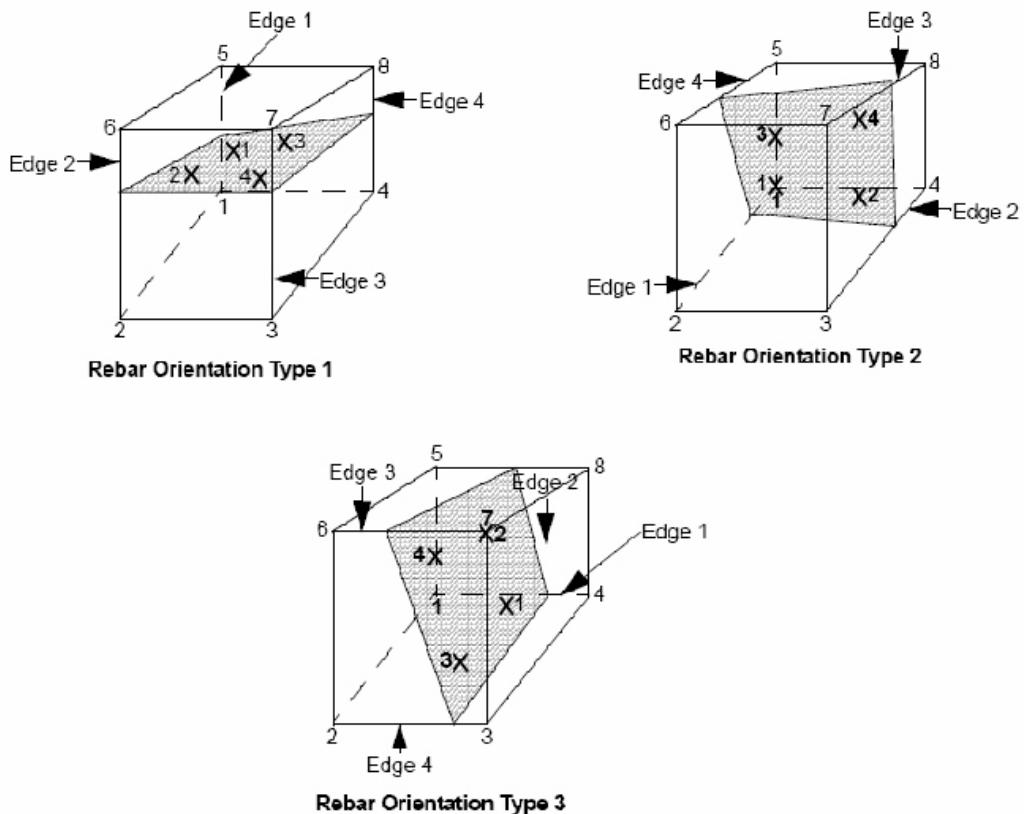
**3-D**

Figure 9-145 Rebar numbering and orientation type (IORI value) - only 3D supported.

**PMREBAR**

Defines Rebar Property Information for CMREBAR Elements in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Enter lines 2 through 6 below to describe up to 5 “rebar layers”. If only the structure only contains two rebar layers, only enter lines 1-3 as shown in the example. SOL 600 only.

**Format:**

| 1       | 2   | 3    | 4     | 5    | 6    | 7      | 8     | 9      | 10 |
|---------|-----|------|-------|------|------|--------|-------|--------|----|
| PMREBAR | IP  |      | IFILE | X1   | X2   | X3     | MICRO | FACTOR |    |
|         | IM1 | POS1 | AREA1 | SSS1 | ANG1 | IPOST1 | IORI1 |        |    |
|         | IM2 | POS2 | AREA2 | SSS2 | ANG2 | IPOST2 | IORI2 |        |    |
|         | IM3 | POS3 | AREA3 | SSS3 | ANG3 | IPOST3 | IORI3 |        |    |
|         | IM4 | POS4 | AREA4 | SSS4 | ANG4 | IPOST4 | IORI4 |        |    |
|         | IM5 | POS5 | AREA5 | SSS5 | ANG5 | IPOST5 | IORI5 |        |    |

**Example (two rebar layers):**

|         |     |       |       |      |         |      |   |      |  |
|---------|-----|-------|-------|------|---------|------|---|------|--|
| PMREBAR | 20  |       | 1     | 1.0  | 0.0     | 0.0  | 1 | 0.01 |  |
|         | 100 | 0.25  | 0.075 | 3.0  | 45.0    | 2000 | 1 |      |  |
|         | 101 | 0.275 | 0.050 | 32.0 | 4 -30.0 | 2001 | 1 |      |  |

**Descriptor      Meaning**

- IP                  Rebar property ID. (Integer; Required; no Default)
- IFILE               Option to create a file for rebar verification. (Integer; Default = 0). If IFILE=1, a file named jid.marc\_rebar.mfd will be created for use by Mentat.
- X1                 First direction cosine of reference axis. (Real; Required; no Default)
- X2                 Second direction cosine of reference axis. (Real; Required; no Default)
- X3                 Third direction cosine of reference axis. (Real; Required; no Default)
- MICRO              Option to activate “micro buckling” behavior. (Integer; Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
- FACTOR             Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real; Default = 0.02)
- IMi                Material identification number. (Integer; Required; no Default)
- POSi               Relative position of rebar layer at edge 1 (pr/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real; Required; no Default)
- AREAi              Rebar cross sectional area. (Real; Required; no Default)

|        |  |
|--------|--|
| SSSi   | Number of rebars per unit length in each layer. (Real; Required; no Default)   |
| ANGi   | Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real; Default = 0.0, must be between -90.0 and 90.0), see <a href="#">Figure 9-146</a> .  |
| IPOSTi | Global identification number of the rebar layer used for postprocessing. (Integer; Default = 0)  |
| IORIi  | Rebar layer orientation type, see <a href="#">Figure 9-147</a> . (Integer; Required; no Default)<br>1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face<br>2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face<br>3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face |

### Remarks:

1. This entry is the property entry for CMREBAR elements and makes use of Marc's REBAR capability for element types 23 and 146. Entries CMREBAI and PMREBAI makes use of Marc's REBAR and INSERT capabilities and use membrane rebar elements 147 and 148.
2. See [Figure 9-146](#) and [Figure 9-147](#) for a definition of ANG and IORI.
3. Any material valid for SOL 600 may be used to define IM.
4. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords).  $\alpha$  is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the [Figure 9-146](#).

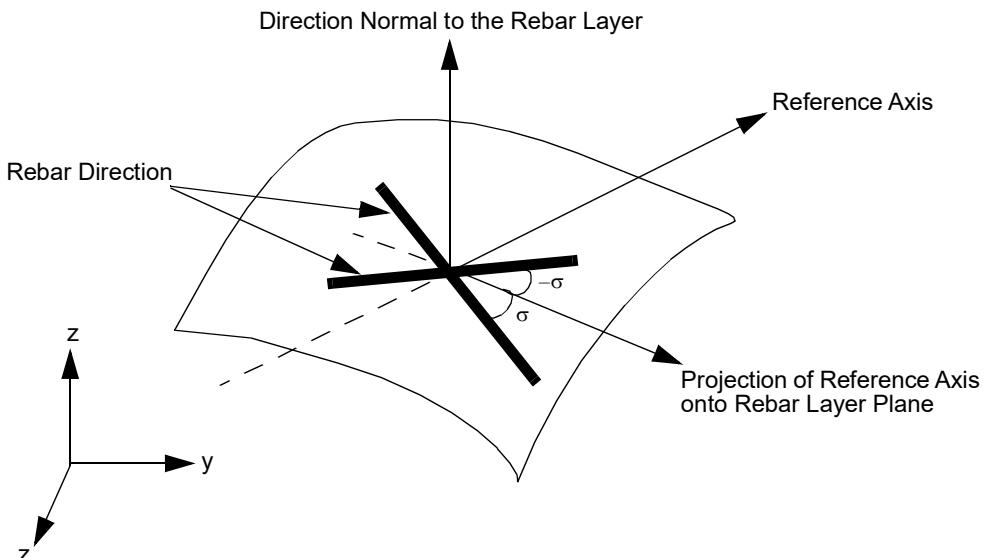


Figure 9-146 Description of rebar orientation on a single rebar layer

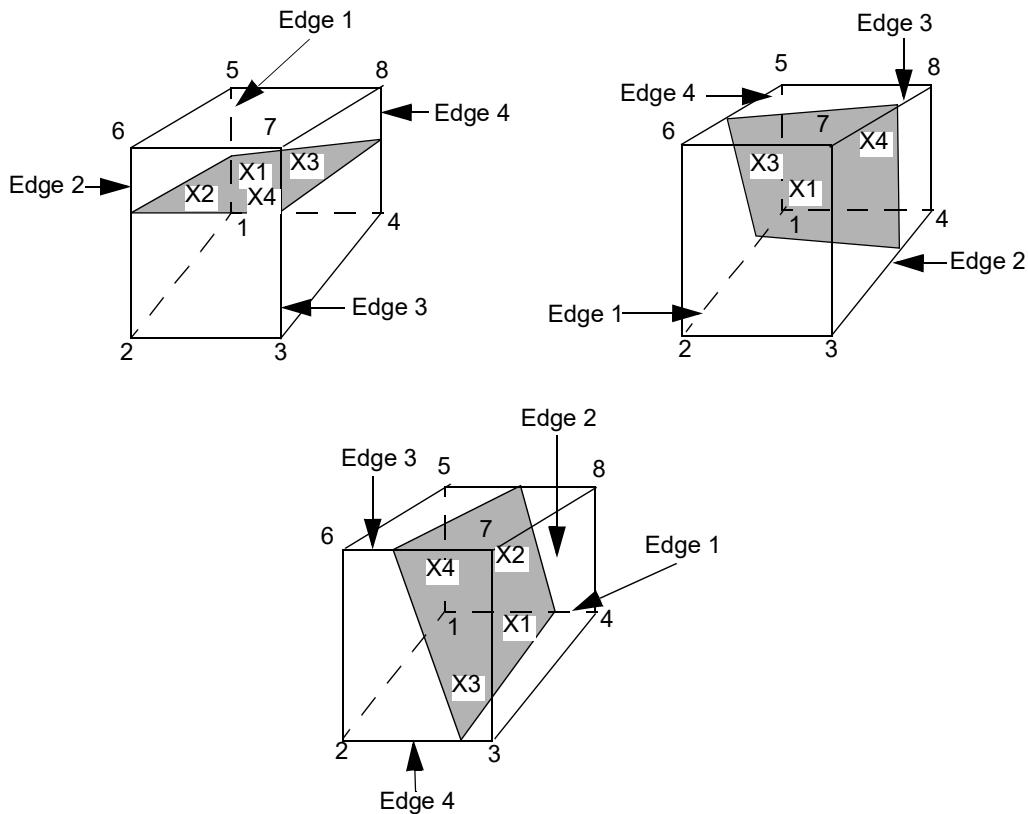


Figure 9-147 Rebar numbering and orientation type (IORI value) - only 3D supported

**POINT****Edge Point for FEEDGE Entry**

Define edge point for FEEDGE or SELOC entries.

**Format:**

| 1     | 2  | 3  | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|-------|----|----|----|----|----|---|---|---|----|
| POINT | ID | CP | X1 | X2 | X3 |   |   |   |    |

**Example:**

|       |    |   |    |    |    |  |  |  |  |
|-------|----|---|----|----|----|--|--|--|--|
| POINT | 12 | 1 | 1. | 2. | 5. |  |  |  |  |
|-------|----|---|----|----|----|--|--|--|--|

| Descriptor | Meaning   | Type             | Default  |
|------------|---|------------------|----------|
| ID         | Point identification number.  | Integer $\geq 0$ | Required |
| CP         | Identification number of coordinate system in which the location of point is defined. | Integer $\geq 0$ | 0        |
| X1, X2, X3 | Location of the point in coordinate system CP.  | Real             | 0.0      |

**Remarks:**

1. POINT is used to specify additional geometric points for edges and can be used by p-version elements. There are no degrees-of-freedom assigned to a point.
2. FEEDGE entries can refer to POINT entries.
3. SELOC entries can refer to POINT entries in the residual or part superelements.
4. ID of POINTs must be unique with respect to ID of GRID entries.
5. POINT entries can be referenced on SET1/SET3 for defining arbitrary beam cross section, ABCS, via PBRSECT/PBMSECT. Note that CP and X3 must be left blank for POINT entries used for ABCS.

**POINTAX****Conical Shell Point**

Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.

**Format:**

| 1       | 2  | 3   | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|-----|-----|---|---|---|---|---|----|
| POINTAX | ID | RID | PHI |   |   |   |   |   |    |

**Example:**

|         |   |   |      |  |  |  |  |  |  |
|---------|---|---|------|--|--|--|--|--|--|
| POINTAX | 2 | 3 | 30.0 |  |  |  |  |  |  |
|---------|---|---|------|--|--|--|--|--|--|

**Descriptor      Meaning**

- |     |  |
|-----|--|
| ID  | Point identification number. (Unique Integer > 0)      |
| RID | Identification number of a RINGAX entry. (Integer > 0) |
| PHI | Azimuthal angle in degrees. (Real)                     |

**Remarks:**

1. This entry is allowed only if an AXIC entry is also present.
2. POINTAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

**PORFCPL****Flow Between Two Coupling Surfaces Through a Hole**

Defines an interaction between two coupling surfaces through a hole. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3    | 4 | 5    | 6    | 7   | 8 | 9 | 10 |
|---------|-----|------|---|------|------|-----|---|---|----|
| PORFCPL | PID | SIZE |   | FLOW | CSID | MID |   |   |    |

**Example:**

|         |   |       |  |      |   |  |  |  |  |
|---------|---|-------|--|------|---|--|--|--|--|
| PORFCPL | 1 | SMALL |  | BOTH | 1 |  |  |  |  |
|---------|---|-------|--|------|---|--|--|--|--|

**Descriptor      Meaning**

PID      Unique PORFCPL identification number. (Integer > 0; Required)

SIZE      Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL)

SMALL      The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.

LARGE      The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.

FLOW      Defines the allowed directions of the flow. (Character; Default = BOTH)

BOTH      In- and outflow are allowed.

IN      Only inflow allowed into the COUPLE that references this entry.

OUT      Only outflow allowed into the COUPLE that references this entry.

CSID      The ID of the COUPLE entry. This COUPLE is the one that is connected to the coupling surface that references this entry. (Integer > 0; Required)

MID      MATDEUL ID number of the transported gas. Only used when connecting to a GBAG and Euler solver uses the multi-material and SIZE=LARGE. See Remark 2. (Integer > 0)

**Remarks:**

1. The PORFCPL entry can only be referenced from LEAKAGE entry.

2. For SIZE=LARGE: once gas from a GBAG flows into an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

**PORFGBG****Flow Between Two Air Bags Through a Hole**

Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3    | 4 | 5    | 6    | 7   | 8 | 9 | 10 |
|---------|-----|------|---|------|------|-----|---|---|----|
| PORFGBG | FID | SIZE |   | FLOW | GBID | MID |   |   |    |

**Example:**

|         |   |       |  |      |   |  |  |  |  |
|---------|---|-------|--|------|---|--|--|--|--|
| PORFGBG | 1 | SMALL |  | BOTH | 1 |  |  |  |  |
|---------|---|-------|--|------|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| FID        | Unique number of a PORFGBG entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or between Eulerian air bag and a GBAG or between Eulerian air bags. (Integer > 0; Required)  |
| SIZE       | Defines the type of flow method that is used for mass leaving or entering the air bag volume. (Character; Default = SMALL)  |
| SMALL      | The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry. |
| LARGE      | The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.   |
| FLOW       | Defines the allowed directions of the flow. (Character; Default = BOTH)   |
| BOTH       | In- and outflow are allowed.  |
| IN         | Only inflow allowed into the GBAG or the coupling surface that references this entry.   |
| OUT        | Only outflow allowed into the GBAG or the coupling surface that references this entry.  |
| GBID       | Number of a GBAG entry. This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)  |
| MID        | MATDEUL ID number of the GBAG gas. Only used when connecting a GBAG to an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 3. (Integer > 0)  |

**Remarks:**

1. The PORFGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler and SIZE=SMALL, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. For SIZE=LARGE: once gas from a GBAG enters an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

**PORFLOW**

## Porous Flow Boundary

Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface. Used in SOL 700 only.

**Format:**

| 1       | 2     | 3      | 4      | 5      | 6     | 7      | 8     | 9      | 10 |
|---------|-------|--------|--------|--------|-------|--------|-------|--------|----|
| PORFLOW | FID   |        | TYPE1  | VALUE1 | TYPE2 | VALUE2 | TYPE3 | VALUE3 |    |
|         | TYPE4 | VALUE4 | -etc.- |        |       |        |       |        |    |

**Example:**

|         |     |  |      |       |   |  |  |  |  |
|---------|-----|--|------|-------|---|--|--|--|--|
| PORFLOW | 120 |  | XVEL | 100.0 | 1 |  |  |  |  |
|         |     |  |      |       |   |  |  |  |  |

**Descriptor      Meaning**

FID              Unique number of a PORFLOW entry. (Integer > 0; Required)

TYPE              The properties on the flow boundary. (Character, Required)

MATERIAL      MATDEUL ID number.

XVEL              Velocity in the x-direction.

YVEL              Velocity in the y-direction.

ZVEL              Velocity in the z-direction.

PRESSURE      Pressure.

DENSITY        Density.

SIE               Specific internal energy.

FLOW              The type of flow boundary required.

METHOD         The method used for the material transport.

VALUEi           The value of the property specified in the TYPE field. (Real or Character, Required)

For TYPEi set to FLOW, the value is a character entry: either IN, OUT or BOTH, indicating that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH. See Remark 4.

For TYPEi set to METHOD, the value is a character entry: either VELOCITY or PRESSURE, indicating that the material transport is based on the velocity method or the pressure method. The default is VELOCITY. See Remark 4.

**Remarks:**

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the (SUB)SURFACE segment is intersecting.

3. The SURFACE can be only a general coupling surface (see the COUPLE entry).
4. The different methods used to calculate the material transport through a porous (sub)surface are described in General Coupling.
5. METHOD=VELOCITY is valid for all equation of state models.  
METHOD=PRESSURE is valid for EOSGAM (ideal gas) in combination with the single material hydrodynamic Euler solver.
6. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
7. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
8. Prescribing both pressure and velocity may lead to the instabilities.

## PORFLWT

Time Dependent Porous Flow Boundary

Defines a time dependent flow through a porous area of the couple surface. Used in SOL 700 only.

### Format:

| 1       | 2       | 3        | 4       | 5      | 6   | 7 | 8 | 9 | 10 |
|---------|---------|----------|---------|--------|-----|---|---|---|----|
| PORFLWT | FID     | TYPE     |         |        |     |   |   |   |    |
|         | VELTYPE | VELOCITY | PRESTYP | PRES   |     |   |   |   |    |
|         | MID     | DENSTYP  | DENSITY | SIETYP | SIE |   |   |   |    |

### Example:

|         |       |       |       |       |     |  |  |  |  |
|---------|-------|-------|-------|-------|-----|--|--|--|--|
| PORFLWT | 2     | IN    |       |       |     |  |  |  |  |
|         | TABLE | 101   | TABLE | 102   |     |  |  |  |  |
|         | 91    | TABLE | 105   | TABLE | 107 |  |  |  |  |

### Descriptor      Meaning

FID      Unique number of a PORFLWT entry. (Integer > 0; Required)

TYPE      Type of flow boundary. (Character; Default = BOTH)

IN      Inflow boundary (see Remarks 2, and 3.).

Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.

OUT      Only outflow is allowed. The outflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use the material mixture as present in the adjacent Euler element.

BOTH      Material is allowed to flow in or out. The switch between inflow and outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be left unspecified. If not given the pressure in the adjacent Euler element will be taken.

VELTYPE      Type of velocity definition. (Character; Default = ELEMENT)

ELEMENT      Value of Euler element.

CONSTANT      Value is constant in time.

TABLE      Value varies in time.

VELOCITY      Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real)

PRESTYP      Type of pressure definition. See Remark 6. (Character)

|         |          |  |
|---------|----------|--|
|         | ELEMENT  | Value of Euler element.  |
|         | CONSTANT | Value is constant in time.   |
|         | TABLE    | Value varies in time.  |
| PRES    |          | Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1. (Integer or Real)   |
| MID     |          | MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and SIE for the inflowing material. (Integer; Default = 0) |
| DENSTYP |          | Type of density definition: (Character, Required when MID > 0.)  |
|         | ELEMENT  | Value of Euler element   |
|         | CONSTANT | Value is constant in time  |
|         | TABLE    | Value varies in time   |
| DENSITY |          | Value of density. If DENSTYP = TABLE, it refers to a TABLED1 ID. (Integer or Real, Required when MID is given.)  |
| SIETYPE |          | Type of density definition. Required when MID is given. (Integer or Real)  |
|         | ELEMENT  | Value of Euler element.  |
|         | CONSTANT | Value is constant in time.   |
|         | TABLE    | Value varies in time.  |
| SIE     |          | Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1. Required when MID is given. (Integer or Real)  |

### Remarks:

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the segment of the coupling surface is intersecting.
3. The surface can be only a general coupling surface (see the COUPLE entry).
4. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
5. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions

6. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be unstable. However, for supersonic inflow one can specify both pressure and velocity since there are no outgoing waves at a supersonic inflow boundary.

**PORHOLE**

Holes in Air Bag Surface

Defines a hole in a COUPLE and/or GBAG surface. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3    | 4 | 5    | 6    | 7      | 8      | 9  | 10 |
|---------|-----|------|---|------|------|--------|--------|----|----|
| PORHOLE | FID | SIZE |   | FLOW | PENV | RHOENV | SIEENV | CP |    |
|         | MID |      |   |      |      |        |        |    |    |

**Example:**

|         |     |  |  |  |  |  |  |  |  |
|---------|-----|--|--|--|--|--|--|--|--|
| PORHOLE | 301 |  |  |  |  |  |  |  |  |
|         |     |  |  |  |  |  |  |  |  |

**Descriptor      Meaning**

|        |   |  |  |  |  |  |  |  |  |
|--------|---|--|--|--|--|--|--|--|--|
| PID    | Unique number of a PORHOLE entry. (Integer > 0; Required)   |  |  |  |  |  |  |  |  |
| SIZE   | Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL)   |  |  |  |  |  |  |  |  |
| SMALL  | The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry. |  |  |  |  |  |  |  |  |
| LARGE  | The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.   |  |  |  |  |  |  |  |  |
| FLOW   | Defines the allowed directions of the flow. (Character; Default = BOTH)   |  |  |  |  |  |  |  |  |
| BOTH   | In- and outflow are allowed.  |  |  |  |  |  |  |  |  |
| IN     | Only inflow allowed.  |  |  |  |  |  |  |  |  |
| OUT    | Only outflow allowed.   |  |  |  |  |  |  |  |  |
| PENV   | Environmental pressure. (Real > 0.0; Required)  |  |  |  |  |  |  |  |  |
| RHOENV | Environmental density. (Real > 0.0; Required)   |  |  |  |  |  |  |  |  |
| SIEENV | Environmental specific internal energy. (Real > 0.0; Required)  |  |  |  |  |  |  |  |  |
| CP     | Environmental specific heat at constant pressure. See Remark 4. (Real > 0.0)  |  |  |  |  |  |  |  |  |
| MID    | MATDEUL ID number of the environment gas. Only used for an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 6. (Integer > 0)   |  |  |  |  |  |  |  |  |

**Remarks:**

1. The PORHOLE entry can be referenced from a LEAKAGE entry.
2. When used with Euler, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. The values for the environment  $p_{env}$  (PENV),  $\rho_{env}$  (RHOENV),  $e_{env}$  (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The  $\gamma_{env}$  is calculated and is only used when inflow occurs. Inflow occurs when  $p_{env} > p_{inside}$ .

4. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.
5. For in and out flow of an uniform pressure air bag (GBAG), the material transport is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is equivalent to the PORHOLE entry.
6. When used in combination with the single material hydrodynamic Euler solver, an EOSGAM (ideal gas) equation of state is required. In that case, the material number, MID, can be left blank. When using the multi-material solver, the material number, MID, has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

**PORHYDS**

## Porous Flow Boundary with a Hydrostatic Pressure Profile

Prescribes a hydrostatic pressure profile on a porous BSURF. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|---|---|---|---|---|---|---|----|
| PORHYDS | PID |   |   |   |   |   |   |   |    |

**Example:**

|         |     |  |  |  |  |  |  |  |  |
|---------|-----|--|--|--|--|--|--|--|--|
| PORHYDS | 120 |  |  |  |  |  |  |  |  |
|---------|-----|--|--|--|--|--|--|--|--|

**Descriptor      Meaning**

PID              Unique number of a PORHYDS entry. (Integer > 0; Required)

**Remarks:**

1. Reference PID by a LEAKAGE entry.
2. The surface can be only a general coupling surface (see the COUPLE entry).
3. It is required that the coupling surface refers to a HYDSTAT entry. This HYDSTAT entry will be used to prescribe a hydrostatic pressure profile on the subsurface. For example, the water level and atmospheric pressure are taken from the HYDSTYAT entry. This defines the pressure and the inflow density.
4. In contributions of the surface to the Euler elements the pressure gradient across the surface is taken into account. Therefore splitting up of the surface and creating new PORHYDS entries does not increase the accuracy of prescribed pressures. If the water level and atmospheric pressure are the same in the whole region outside the coupling surface using one PORHYDS entry is sufficient.
5. The atmospheric pressure is prescribed on those parts of the surface that are above the water level.

**PORUDS**

User-defined Porosity Model Specified by a User Defined Service.

Defines a porosity model of a COUPLE surface through a user-written subroutine. Use in SOL700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|---|---|---|---|---|----|
| PORUDS | PID | GROUP | UNAME |   |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE mypor 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

*In Bulk Data:*

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| PORUDS | 12 | mypor | EXPOR |   |   |   |   |   |    |

| Descriptor | Meaning  |
|------------|--|
| PID        | Unique porosity model ID. (Integer > 0; Required)  |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXPOR)                 |

**Remarks:**

1. Only can be used for SOL 700.
2. The porosity ID (PID) must be referenced by a LEAKAGE entry.
3. UNAME can be:

| Subroutine Name | Function  |
|-----------------|---|
| EXPOR           | Standard user defined flow boundary on the coupling |

**PRAC2D****CRAC2D Element Property**

Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.

**Format:**

| 1      | 2   | 3   | 4 | 5      | 6   | 7     | 8   | 9 | 10 |
|--------|-----|-----|---|--------|-----|-------|-----|---|----|
| PRAC2D | PID | MID | T | IPLANE | NSM | GAMMA | PHI |   |    |

**Example:**

|        |     |   |      |   |     |     |      |  |  |
|--------|-----|---|------|---|-----|-----|------|--|--|
| PRAC2D | 108 | 2 | 0.10 | 0 | .17 | .50 | 180. |  |  |
|--------|-----|---|------|---|-----|-----|------|--|--|

**Describer      Meaning**

|        |  |
|--------|--|
| PID    | Property identification number. (Integer > 0)  |
| MID    | Material identification number. (Integer > 0)  |
| T      | Element thickness. (Real > 0.0)  |
| IPLANE | Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress. (Integer = 0 or 1, Default = 0)                                   |
| NSM    | Non-structural mass per unit area. (Real $\geq$ 0.0; Default = 0)  |
| GAMMA  | Exponent used in the displacement field. See Remark 4. (Real; Default = 0.5)   |
| PHI    | Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated. See Remark 4. (Real; Default = 180.0) |

**Remarks:**

1. PRAC2D is a primary property entry. Primary property entries are grouping entities for many applications in Nastran. Therefore it is highly recommended that the PRAC2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC2D entries.
2. PRAC2D entry may refer to MAT1, MAT2, or MAT8 material property entries.
3. For plane strain analysis, only MAT1 type data should be used.
4. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

**PRAC3D****CRAC3D Element Property**

Defines the properties of the CRAC3D structural element.

**Format:**

| 1      | 2   | 3   | 4     | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|-------|-----|---|---|---|---|----|
| PRAC3D | PID | MID | GAMMA | PHI |   |   |   |   |    |

**Example:**

|        |     |   |     |      |  |  |  |  |  |
|--------|-----|---|-----|------|--|--|--|--|--|
| PRAC3D | 108 | 2 | .50 | 180. |  |  |  |  |  |
|--------|-----|---|-----|------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0)  |
| MID        | Material identification number. (Integer > 0)  |
| GAMMA      | Exponent used in the displacement field. See Remark 3. (Real; Default = 0.5)   |
| PHI        | Angle (in degrees) relative to the element x axis along which stress intensity factors are to be calculated. See Remark 3. (Real; Default = 180.0) |

**Remarks:**

1. PRAC3D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PRAC3D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC3D entries.
2. Either isotropic (MAT1) or anisotropic (MAT9) material entries may be referenced.
3. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

**PRESAX**

## Conical Shell Pressure Load

Defines the static pressure loading on a conical shell element.

**Format:**

| 1      | 2   | 3 | 4    | 5    | 6    | 7    | 8 | 9 | 10 |
|--------|-----|---|------|------|------|------|---|---|----|
| PRESAX | SID | P | RID1 | RID2 | PHI1 | PHI2 |   |   |    |

**Example:**

|        |   |      |   |   |      |      |  |  |  |
|--------|---|------|---|---|------|------|--|--|--|
| PRESAX | 3 | 7.92 | 4 | 3 | 20.6 | 31.4 |  |  |  |
|--------|---|------|---|---|------|------|--|--|--|

**Describer      Meaning**

|            |  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)                |
| P          | Pressure value. (Real)                                       |
| RID1, RID2 | Ring identification numbers. See RINGAX entry. (Integer > 0) |
| PHI1, PHI2 | Azimuthal angles in degrees. (Real; PHI2 > PHI1)             |

**Remarks:**

1. PRESAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD = SID.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
4. For axisymmetric loading over 360 degrees, use PHI1 = 0.0 and PHI2 = 360.0.

**PRESPT**

## Fluid Pressure Point

Defines the location of pressure points in the fluid for recovery of pressure data.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6    | 7    | 8    | 9 | 10 |
|--------|-----|------|------|------|------|------|------|---|----|
| PRESPT | IDF | IDP1 | PHI1 | IDP2 | PHI2 | IDP3 | PHI3 |   |    |

**Example:**

|        |    |     |     |  |  |     |      |  |  |
|--------|----|-----|-----|--|--|-----|------|--|--|
| PRESPT | 14 | 141 | 0.0 |  |  | 142 | 90.0 |  |  |
|--------|----|-----|-----|--|--|-----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDF        | Fluid point (RINGFL entry) identification number. (Integer > 0)                        |
| IDPi       | Pressure point identification number. (Integer > 0)                                    |
| PHIi       | Azimuthal position on fluid point referenced by IDF in fluid coordinate system. (Real) |

**Remarks:**

1. PRESPT is allowed only if an AXIF entry is also present.
2. All pressure point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The pressure points are used primarily for the identification of output data. They may also be used as points at which to measure pressure for input to control devices.
4. One, two, or three pressure points may be defined per entry.
5. Output requests for velocity and acceleration of these degrees-of-freedom will result in derivatives of pressure with respect to time.

**PRIMx**

## Thermal Geometric Primitives for RC Radiation

Specifies the properties of geometric primitives to be used in radiation calculations in place of elements.

**Format (GEOM2):**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8 | 9 | 10 |
|-------|--------|--------|--------|---------|---------|--------|---|---|----|
| PRIM1 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |   |   |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |   |   |    |
|       | P3(1)  | P3(2)  | P3(3)  | A_mesh  | B_mesh  |        |   |   |    |

**Example:**

|       |    |     |     |    |    |    |  |  |  |
|-------|----|-----|-----|----|----|----|--|--|--|
| PRIM1 | 11 | 101 | 102 | 3  | 4  | 2  |  |  |  |
|       | 0. | 0.  | 0.  | 1. | 0. | 0. |  |  |  |
|       | 0. | 1.  | 0.  | 3  | 4  |    |  |  |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9      | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|--------|----|
| PRIM2 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |        |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |        |    |
|       | P3(1)  | P3(2)  | P3(3)  | P4(1)   | P4(2)   | P4(3)  | A_mesh | B_mesh |    |

**Example:**

|       |    |     |     |    |    |    |   |   |  |
|-------|----|-----|-----|----|----|----|---|---|--|
| PRIM2 | 12 | 102 | 103 | 3  | 4  | 2  |   |   |  |
|       | 0. | 0.  | 0.  | 1. | 0. | 0. |   |   |  |
|       | 0. | 1.  | 0.  | 1. | 1. | 0. | 3 | 4 |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8 | 9 | 10 |
|-------|--------|--------|--------|---------|---------|--------|---|---|----|
| PRIM3 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |   |   |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |   |   |    |
|       | P3(1)  | P3(2)  | P3(3)  | A_mesh  | B_mesh  |        |   |   |    |

**Example:**

|       |    |     |     |    |    |    |  |  |  |
|-------|----|-----|-----|----|----|----|--|--|--|
| PRIM3 | 13 | 103 | 104 | 3  | 4  | 2  |  |  |  |
|       | 0. | 0.  | 0.  | 1. | 0. | 0. |  |  |  |
|       | 0. | 1.  | 0.  | 3  | 4  |    |  |  |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9 | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|---|----|
| PRIM4 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |   |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |   |    |
|       | P3(1)  | P3(2)  | P3(3)  | Diam1   | Diam2   | Angle1 | Angle2 |   |    |
|       | A_mesh | B_mesh |        |         |         |        |        |   |    |

**Example:**

|       |    |     |     |     |     |     |      |  |  |
|-------|----|-----|-----|-----|-----|-----|------|--|--|
| PRIM4 | 14 | 104 | 105 | 3   | 4   | 2   |      |  |  |
|       | 0. | 0.  | 0.  | 0.0 | 1.0 | 0.  |      |  |  |
|       | 0. | 0.  | 1.  | 1.  | 0.  | 60. | 180. |  |  |
|       | 3  | 4   |     |     |     |     |      |  |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9      | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|--------|----|
| PRIM5 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |        |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |        |    |
|       | P3(1)  | P3(2)  | P3(3)  | Diam1   | Angle1  | Angle2 | A_mesh | B_mesh |    |

**Example:**

|       |    |     |     |    |     |      |   |   |  |
|-------|----|-----|-----|----|-----|------|---|---|--|
| PRIM5 | 15 | 105 | 106 | 3  | 4   | 2    |   |   |  |
|       | 0. | 0.  | 0.  | 0. | 1.  | 0.   |   |   |  |
|       | 0. | 0.  | 1.  | 1. | 60. | 180. | 3 | 4 |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9 | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|---|----|
| PRIM6 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |   |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |   |    |
|       | P3(1)  | P3(2)  | P3(3)  | Diam1   | Diam2   | Angle1 | Angle2 |   |    |
|       | A_mesh | B_mesh |        |         |         |        |        |   |    |

**Example:**

|       |    |     |     |    |    |     |     |  |  |
|-------|----|-----|-----|----|----|-----|-----|--|--|
| PRIM6 | 16 | 106 | 107 | 3  | 4  | 2   |     |  |  |
|       | 0. | 0.  | 0.  | 0. | 1. | 0.  |     |  |  |
|       | 0. | 0.  | 1.  | 1. | 0. | 60. | 80. |  |  |
|       | 3  | 4   |     |    |    |     |     |  |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9      | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|--------|----|
| PRIM7 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |        |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |        |    |
|       | P3(1)  | P3(2)  | P3(3)  | Diam1   | Angle1  | Angle2 | Trunc1 | Trunc2 |    |
|       | A_mesh | B_mesh |        |         |         |        |        |        |    |

**Example:**

|       |    |     |     |    |     |      |      |     |  |
|-------|----|-----|-----|----|-----|------|------|-----|--|
| PRIM7 | 17 | 107 | 108 | 3  | 4   | 2    |      |     |  |
|       | 0. | 0.  | 0.  | 0. | 1.  | 0.   |      |     |  |
|       | 0. | 0.  | 1.  | 1. | 60. | 180. | -0.5 | 0.5 |  |
|       | 3  | 4   |     |    |     |      |      |     |  |

**Format:**

| 1     | 2      | 3      | 4      | 5       | 6       | 7      | 8      | 9      | 10 |
|-------|--------|--------|--------|---------|---------|--------|--------|--------|----|
| PRIM8 | PRIMID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |        |        |    |
|       | P1(1)  | P1(2)  | P1(3)  | P2(1)   | P2(2)   | P2(3)  |        |        |    |
|       | P3(1)  | P3(2)  | P3(3)  | Diam1   | Angle1  | Angle2 | Trunc1 | Trunc2 |    |
|       | A_mesh | B_mesh |        |         |         |        |        |        |    |

**Example:**

|       |    |     |     |    |     |      |     |    |  |
|-------|----|-----|-----|----|-----|------|-----|----|--|
| PRIM8 | 18 | 108 | 109 | 3  | 4   | 2    |     |    |  |
|       | 0. | 0.  | 0.  | 0. | 1.  | 0.   |     |    |  |
|       | 0. | 0.  | 1.  | 1. | 60. | 180. | 0.5 | 1. |  |
|       | 3  | 4   |     |    |     |      |     |    |  |

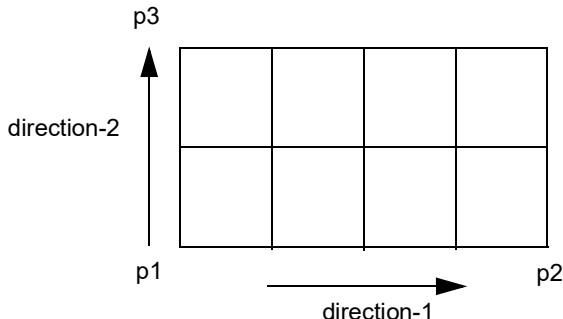
| Descriptor | Meaning  |
|------------|--|
| PRIMID     | Primitive identification number, unique to all other PRIMx (Integer > 0; Required)   |
| SET3 ID    | ID of the element collection to which this primitive describes. This collection acts as the elements that will exist in the thermal model, but the collection will be absent from the radiation model. Instead, the primitive will be used to calculate radiation, and be redistributed back onto the elements. All radiation properties for the primitive will be applied to the element collection, and must be consistent across. (Integer > 0; Required) |
| RADMID     | ID of the radiation material properties used to describe this primitive for analysis.  |
| Pi(a)      | The position of point i in the a axis, as described in the correlating picture. For example P2(2) denotes the y coordinate of the second point. Position is always described in global coordinates. (Real; Required)   |

| Descriptor | Meaning   |
|------------|---|
| Diamx      | Diameter x of the primitive, if applicable and as described in the correlating picture.<br>(Real $\geq 0.0$ ; Required)   |
| Anglex     | Angle x of the primitive in degrees, if applicable and as described in the correlating picture.<br>( $0 \leq \text{Real} \leq 360.0$ ; Required)  |
| Truncx     | Truncation x of the primitive, if applicable and as described in the correlating picture. (For PRIM8, Real $\geq 0$ ; Required). (For PRIM7, $-0.5 * \text{Diam1} \leq \text{Real} \leq 0.5 * \text{Diam1}$ ) |
| A_mesh     | Number of mesh spaces in parametric direction-1, as described in the correlating picture.<br>(Integer $> 0$ ; Required)   |
| B_mesh     | Number of mesh spaces in parametric direction-2, as described in the correlating picture.<br>(Integer $> 0$ ; Required)   |

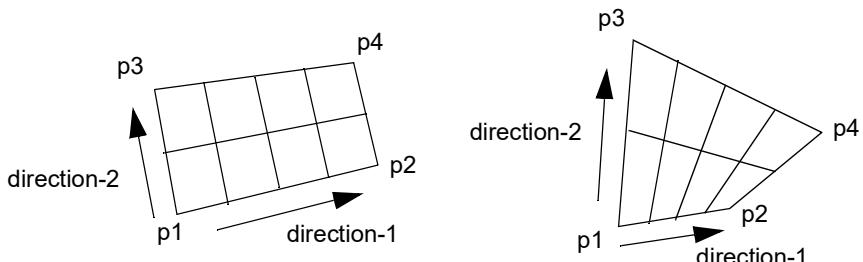
**Remarks:**

1. This entry is for RC Network solver only.
2. Set 3 should include all the elements which belong to this primitive. It will cause wrong results if only partial of the elements are included.
3. About the primitives

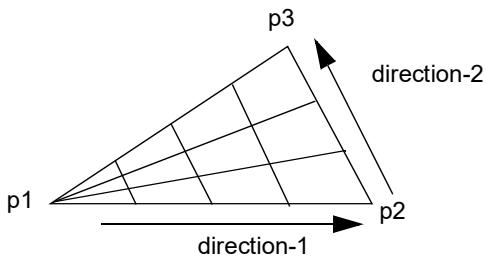
Prim1: Rectangle



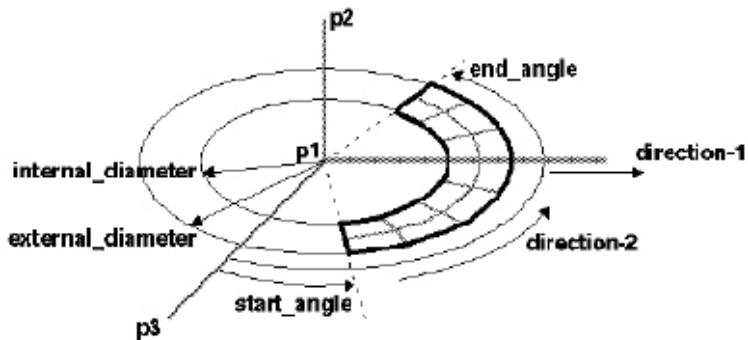
Prim2: Quad



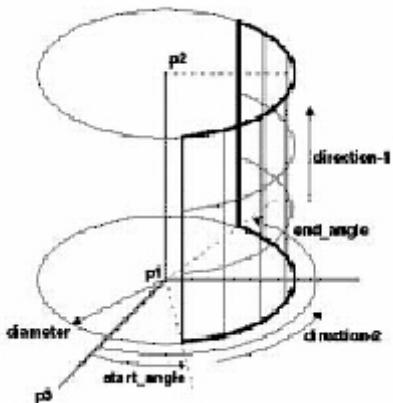
Prim3: Triangle



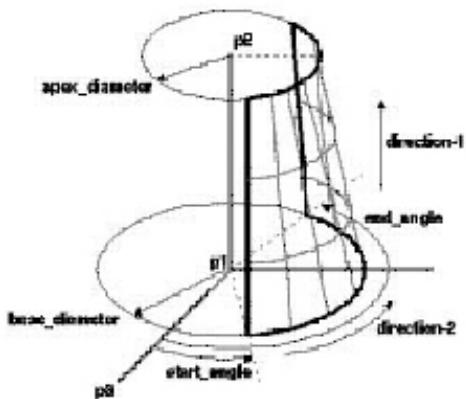
Prim4: Disc



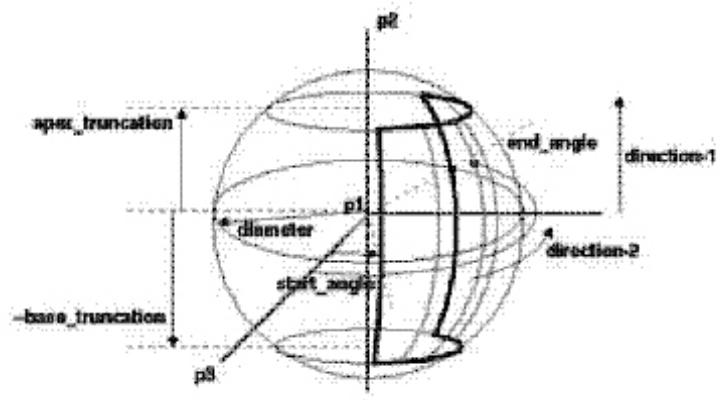
Prim5: Cylinder



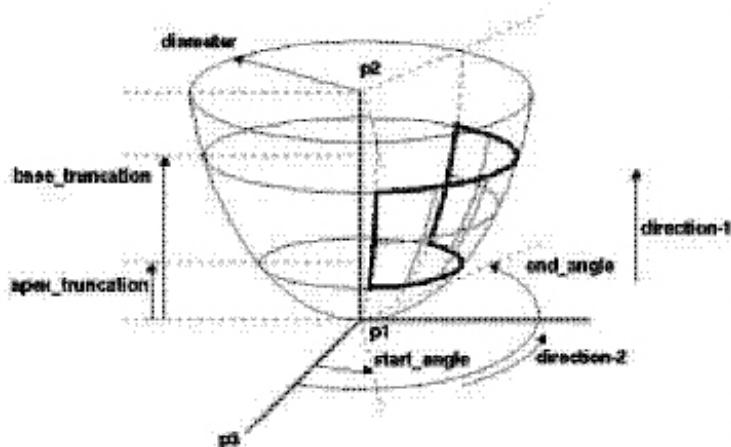
Prim6: Cone



Prim7: Sphere



## Prim8: Parabolic



**PRJCON****Thermal RC Element Contact**

Specifies a thermal connection between two regions of elements. The connection is automatically determined geometrically as a projection of the slave region on to the master, and the strength of the connection is calculated based on the properties given.

**Format: (HEAT1)**

| 1      | 2       | 3              | 4          | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---------|----------------|------------|---|---|---|---|---|----|
| PRJCON | BID     |                |            |   |   |   |   |   |    |
|        | "HEAT1" | SET3<br>MASTER | SET3 SLAVE | h |   |   |   |   |    |

**Example:**

|        |       |   |   |     |  |  |  |  |  |
|--------|-------|---|---|-----|--|--|--|--|--|
| PRJCON | 1     |   |   |     |  |  |  |  |  |
|        | HEAT1 | 1 | 2 | 1.2 |  |  |  |  |  |

**Format: (HEAT2)**

| 1      | 2       | 3              | 4          | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|---------|----------------|------------|----|---|---|---|---|----|
| PRJCON | BID     |                |            |    |   |   |   |   |    |
|        | "HEAT2" | SET3<br>MASTER | SET3 SLAVE | ID |   |   |   |   |    |

**Example:**

|        |       |   |   |      |  |  |  |  |  |
|--------|-------|---|---|------|--|--|--|--|--|
| PRJCON | 1     |   |   |      |  |  |  |  |  |
|        | HEAT2 | 1 | 2 | 1001 |  |  |  |  |  |

**Format: (HEAT3)**

| 1      | 2       | 3              | 4          | 5 | 6           | 7          | 8 | 9 | 10 |
|--------|---------|----------------|------------|---|-------------|------------|---|---|----|
| PRJCON | BID     |                |            |   |             |            |   |   |    |
|        | "HEAT3" | SET3<br>MASTER | SET3 SLAVE | F | Emis Master | Emis Slave |   |   |    |

**Example:**

|        |       |   |   |    |      |     |  |  |  |
|--------|-------|---|---|----|------|-----|--|--|--|
| PRJCON | 1     |   |   |    |      |     |  |  |  |
|        | HEAT3 | 1 | 2 | 1. | 0.85 | 0.5 |  |  |  |

**Format: (HEAT4)**

| 1      | 2       | 3              | 4          | 5 | 6                 | 7                | 8 | 9 | 10 |
|--------|---------|----------------|------------|---|-------------------|------------------|---|---|----|
| PRJCON | BID     |                |            |   |                   |                  |   |   |    |
|        | "HEAT4" | SET3<br>MASTER | SET3 SLAVE | F | RADC ID<br>Master | RADC ID<br>Slave |   |   |    |

**Example:**

|        |       |   |   |    |      |      |  |  |  |
|--------|-------|---|---|----|------|------|--|--|--|
| PRJCON | 1     |   |   |    |      |      |  |  |  |
|        | HEAT4 | 1 | 2 | 1. | 1001 | 1002 |  |  |  |

| Descriptor     | Meaning   |
|----------------|---|
| BID            | BCBODY identification number (Integer > 0; Required)                                      |
| HEATx          | Indicates the start of HEAT elements (Character)  |
| SET3 Master    | ID of the master element collection for connection (Integer > 0; Required)                |
| SET3 Slave     | ID of the slave element collection for connection (Integer > 0; Required)                 |
| h              | Convection correlation (Real $\geq 0.0$ ; Required for HEAT1)                             |
| PCONID         | PCONID of the property to be used for h value. (Integer > 0; Required for HEAT2)          |
| F              | View factor between parts ( $0.0 \leq \text{Real} \leq 1.0$ ; Required for HEAT3 & HEAT4) |
| Emis Master    | Emissivity of master collection (Real $\geq 0.0$ ; Required for HEAT3)                    |
| Emis Slave     | Emissivity of slave collection (Real $\geq 0.0$ ; Required for HEAT3)                     |
| RADC id Master | RADMID of the material to be used for master emissivity value. (Integer > 0; Required)    |
| RADC id Slave  | RADMID of the material to be used for slave emissivity value. (Integer > 0; Required)     |

**Remarks:**

1. This “HEATx” and the later parameters are for RC Network solver only.
2. For HEAT2, PCONID must refer to a PCONV1 type and not a PCONV or PCONVM.
3. RC Network Solver uses a projection method to determine the connection (not the “nearest neighbor method”). In most of the cases, the projection method is more accurate than the nearest neighbor method.

**PROD****Rod Property**

Defines the properties of a rod element (CROD entry).

**Format:**

| 1    | 2   | 3   | 4 | 5 | 6 | 7   | 8 | 9 | 10 |
|------|-----|-----|---|---|---|-----|---|---|----|
| PROD | PID | MID | A | J | C | NSM |   |   |    |

**Example:**

|      |    |    |      |       |        |     |  |  |  |
|------|----|----|------|-------|--------|-----|--|--|--|
| PROD | 17 | 23 | 42.6 | 17.92 | 4.2356 | 0.5 |  |  |  |
|------|----|----|------|-------|--------|-----|--|--|--|

| <b>Describer</b> | <b>Meaning</b>  |
|------------------|---|
| PID              | Property identification number. (Integer > 0)                       |
| MID              | Material identification number. See Remarks 2. and 3. (Integer > 0) |
| A                | Area of the rod. (Real)   |
| J                | Torsional constant. (Real)  |
| C                | Coefficient to determine torsional stress. (Real; Default = 0.0)    |
| NSM              | Nonstructural mass per unit length. (Real)                          |

**Remarks:**

1. PROD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PROD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PROD entries.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a reference MAT4 or MAT5 entry.
4. The formula used to calculate torsional stress is

$$\tau = \frac{CM_0}{J}$$

where  $M_0$  is the torsional moment.

**PRODN1**

## Nonlinear Property Extensions for a PROD Entry

Specifies additional nonlinear properties for elements that point to a PROD entry in SOL 400.

**Format:**

| 1      | 2    | 3    | 4    | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|------|------|------|-------|-------|---|---|---|----|
| PRODN1 | PID  | MID  |      |       | ANAL  |   |   |   |    |
|        | “C2” | BEH2 | INT2 | BEH2H | INT2H |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PRODN1 | 22 | 98 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

**Descriptor      Meaning**

|       |   |
|-------|---|
| PID   | Property identification number of an existing PROD entry. (Integer > 0)   |
| MID   | Material identification number. Remarks 7.(Integer $\geq 0$ )   |
| ANAL  | Analysis type. ANAL='IS' - Implicit structural elements are being referred to.<br>ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL='ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH). |
| C2    | Keyword indicating that items following apply to elements with two end grids.<br>(Character)  |
| BEH2  | Element structural behavior. See Remark 4. (Character Default ROD)  |
| INT2  | Integration scheme. See Remarks 4. and 5. (Character Default L)   |
| BEH2H | Element heat behavior. See Remark 4. (Character Default ROD)  |
| INT2H | Integration scheme. See Remarks 4. and 5. (Character Default L)   |

**Remarks:**

1. The PID above must point to an existing PROD Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PROD entry. If > 0 it will override the MID value on the PROD.
3. The MID entry may point to the MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

**Caution:** The MATVE, MATVP, MATEP, and MATF entries are only associated with a CROD element if the CROD element refers to a PRODN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

| Implicit Structural Materials |  |
|-------------------------------|--|
| MAT1                          |  |
| MATVE                         |  |
| MATVP                         |  |
| MATEP                         |  |
| MATF                          |  |
| MATS1                         |  |

| Heat Materials |  |
|----------------|--|
| MAT4           |  |
| MAT5           |  |

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness of the MAT4 and MAT5 entries.

4. BEH2/BEH2H refers to the nonlinear structural/heat behavior of the ROD element. An underlined item delineates default.

| Structural/Heat Classification of Elements |                 |                  |              |         |
|--|-----------------|------------------|--------------|---------|
| Element Structural/Heat Type               | BEH2/BEH2H CODE | Integration Code | Element Type | # Nodes |
| Rod  | ROD             | L                | ROD          | 2       |

5. Integration codes in Remark 4. are:

| INT CODE | Integration Type |
|----------|------------------|
| L        | Linear           |

6. Any J, C, or NSM value on the PROD will be ignored.
7. The structural element damping coefficient, GE, is not supported on elements which reference PRODN1.

**PSEAM****CSEAM Property**

Defines the PSEAM property values.

**Format:**

| 1     | 2   | 3   | 4    | 5 | 6 | 7  | 8 | 9 | 10 |
|-------|-----|-----|------|---|---|----|---|---|----|
| PSEAM | PID | MID | TYPE | W | T | IN |   |   |    |

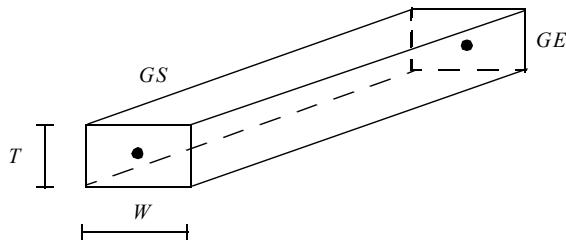
**Example:**

|       |   |   |  |     |  |  |  |  |  |
|-------|---|---|--|-----|--|--|--|--|--|
| PSEAM | 7 | 1 |  | 16. |  |  |  |  |  |
|-------|---|---|--|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number. (Integer > 0)  |
| MID        | Material identification number. (Integer > 0)  |
| TYPE       | “KEYWORD” type of Seam Weld generated. (Character; Default = LINE)   |
| W          | Width of the SEAM. See Remark 1. (Real > 0.)   |
| T          | Thickness of the SEAM. See Remark 2. (Real > 0. or blank)  |
| IN         | Integration scheme. See Remark 6. of PSOLID entry. (Integer 0 or 2; Default = 2)<br>If IN = 0, 2x2x2 reduced shear integration with bubble functions.<br>If IN = 2, 2x2x2 reduced shear integration. |

**Remarks:**

1. The length of the SEAM is the distance between GS and GE. The width W of the SEAM is measured perpendicular to the length and lies in the plane of the patches A and B (see [Figure 9-148](#)). The width is also used to find the projection of the SEAM on the two patches A and B.



**Figure 9-148** Dimensions of a CSEAM Element

2. If left blank, the thickness will be computed as  $T = (T_A + T_B)/2$  where  $T_A$  is the thickness of patch A and  $T_B$  is the thickness of patch B.

3. PSEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSEAM entries.

**PSET****p-Version Element Polynomial Distribution**

Describes polynomial order distribution and is selected by the ADAPT Case Control command.

**Format:**

| 1    | 2   | 3     | 4     | 5     | 6   | 7      | 8  | 9 | 10 |
|------|-----|-------|-------|-------|-----|--------|----|---|----|
| PSET | SID | POLY1 | POLY2 | POLY3 | CID | SETTYP | ID |   |    |

**Example:**

|      |     |   |   |   |  |  |    |  |  |
|------|-----|---|---|---|--|--|----|--|--|
| PSET | 127 | 1 | 2 | 1 |  |  | 12 |  |  |
|------|-----|---|---|---|--|--|----|--|--|

| Descriptor | Meaning  | Type             | Default   |
|------------|--|------------------|-----------|
| SID        | ID selected in the ADAPT Case Control command.   | Integer > 0      | Required  |
| CID        | Coordinate system used to specify polynomial values in different directions. See Remark 1. | Integer $\geq 0$ | Remark 2. |
| POLYi      | Polynomial order in 1, 2, 3 directions of the CID system.                                  | Integer > 0      | Remark 3. |
| SETTYP     | Type of set provided ("SET" or "ELID")   | Character        | "SET"     |
| ID         | SET ID or element ID with this p-value specification.                                      | Integer > 0      | 999999    |

**Remarks:**

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element's topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the midpoint of this vector.) The p-level of each edge is now determined by the nearest integer to

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where  $(n_1, n_2, n_3)$  are the components of this unit vector in the CID system.

3. The default value for POLY2 and POLY3 is POLY1.
4. Any overlap of the PSET specification will result in a warning message and the use of the PSET with the highest pi entry.

5. Whenever SETTYP = "SET", a SET command must be defined in the SETS DEFINITION section of the Case Control Section.
6. SET = 999999 is a reserved set that includes all elements.
7. Whenever there are more than one PSET entries for a given element, then:
  - If CID on the PSET entries are the same, the entry with the maximum POLYi will be used.
  - If CID on the PSET entries are different, a fatal message is issued.

**PSHEAR****Shear Panel Property**

Defines the properties of a shear panel (CSHEAR entry).

**Format:**

| 1      | 2   | 3   | 4 | 5   | 6  | 7  | 8 | 9 | 10 |
|--------|-----|-----|---|-----|----|----|---|---|----|
| PSHEAR | PID | MID | T | NSM | F1 | F2 |   |   |    |

**Example:**

|        |    |    |      |       |       |     |  |  |  |
|--------|----|----|------|-------|-------|-----|--|--|--|
| PSHEAR | 17 | 23 | 42.6 | 17.92 | 4.236 | 0.5 |  |  |  |
|--------|----|----|------|-------|-------|-----|--|--|--|

**Descriptor      Meaning**

- |     |  |
|-----|--|
| PID | Property identification number. (Integer > 0)  |
| MID | Material identification number of a MAT1 entry. (Integer > 0)  |
| T   | Thickness of shear panel. (Real ≠ 0.0)   |
| NSM | Nonstructural mass per unit area. (Real)   |
| F1  | Effectiveness factor for extensional stiffness along edges 1-2 and 3-4. See Remark 2.<br>(Real ≥ 0.0; Default = 0.0) |
| F2  | Effectiveness factor for extensional stiffness along edges 2-3 and 1-4. See Remark 2.<br>(Real ≥ 0.0; Default = 0.0) |

**Remarks:**

1. PSHEAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSHEAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHEAR entries.
2. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If  $F1 \leq 1.01$ , the areas of the rods on edges 1-2 and 3-4 are set equal to  $(F1 \cdot T \cdot PA)/(L12 + L34)$  where PA is the panel surface area-half the vector cross product area of the diagonals-and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if  $F1 = 1.0$ , the panel is fully effective for extension in the 1-2 direction. If  $F1 > 1.01$ , the areas of the rods on edges 1-2 and 3-4 are each set equal to  $0.5 \cdot F1 \cdot T^2$ .

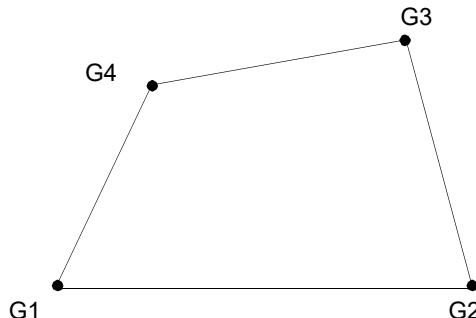


Figure 9-149 Extensional Area for Shear Panel

Thus, if  $F_1 = 30$ , the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to  $15T$ . The significance of  $F_2$  for edges 2-3 and 1-4 is similar.

3. Poisson's ratio coupling for extensional effects is ignored.
4. The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: See Remark 4. of the CSHEAR entry.

**PSHEARN**

## Nonlinear Property Extensions for a PSHEAR Entry

Specifies nonlinear properties for elements that point to a PSHEAR entry in SOL 400.

**Format:**

| 1       | 2    | 3    | 4    | 5 | 6    | 7 | 8 | 9 | 10 |
|---------|------|------|------|---|------|---|---|---|----|
| PSHEARN | PID  | MID  |      |   | ANAL |   |   |   |    |
|         | "C4" | BEH4 | INT4 |   |      |   |   |   |    |

**Example:**

|         |    |    |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|
| PSHEARN | 22 | 18 |  |  |  |  |  |  |  |
|---------|----|----|--|--|--|--|--|--|--|

**Descriptor      Meaning**

|      |   |
|------|---|
| PID  | Property identification number of an existing PSHEAR entry. (Integer > 0)                                   |
| MID  | Material identification number. Remark 9. (Integer $\geq 0$ or blank)                                       |
| ANAL | Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to.<br>(Character: Default IS) |
| C4   | Keyword indicating that two items following apply to elements with four corner grids.<br>(Character)        |
| BEH4 | Element structural behavior. See Remark 5. (Character Default MB)   |
| INT4 | Integration scheme. See Remarks 4. and 6. (Character Default L)   |

**Remarks:**

1. The PID must point to an existing PSHEAR Bulk Data entry and is honored only in SOL 400.
2. Only large membrane rotation is supported. Stringer effectiveness is ignored and only membrane action is considered.
3. MID if blank (or 0) use the MID value on the PSHEAR entry. If > 0 is will override the MID value on the PSHEAR.
4. The MID entry for nonlinear structures may point to the MAT entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The MATVE, MATVP, MATEP, MATF and MATSMA entries are only associated with a CSHEAR element if the CSHEAR element refers to a PSHEARN entry.

| Implicit Structure Materials |
|------------------------------|
| MAT1                         |
| MATVE                        |
| MATVP                        |
| MATEP                        |
| MATF                         |
| MATS1                        |
| MATSMA                       |

5. BEH4 refers to the nonlinear structural behavior of the SHEAR element. An underlined item delineates default.

| Structural Classification of Elements |                 |                  |              |         |
|---------------------------------------|-----------------|------------------|--------------|---------|
| Element Structural Type               | BEH2/BEH2H CODE | Integration Code | Element Type | # Nodes |
| SHEAR                                 | MB              | <u>L</u>         | SHEAR        | 4       |

6. Integration codes in Remark 4. are:

| INT CODE | Integration Type |
|----------|------------------|
| L        | Linear           |

7. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
8. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
9. The structural element damping coefficient, GE, is not supported on elements which reference PSHEARN.

**PSHELL****Shell Element Property**

Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.

**Format:**

| 1      | 2   | 3    | 4    | 5    | 6             | 7    | 8    | 9   | 10 |
|--------|-----|------|------|------|---------------|------|------|-----|----|
| PSHELL | PID | MID1 | T    | MID2 | $12I/T^{**3}$ | MID3 | TS/T | NSM |    |
|        | Z1  | Z2   | MID4 |      |               |      |      |     |    |

**Example:**

|        |     |      |      |     |     |     |     |      |  |
|--------|-----|------|------|-----|-----|-----|-----|------|--|
| PSHELL | 203 | 204  | 1.90 | 205 | 1.2 | 206 | 0.8 | 6.32 |  |
|        | .95 | -.95 |      |     |     |     |     |      |  |

| Descriptor    | Meaning   |
|---------------|---|
| PID           | Property identification number. (Integer > 0)   |
| MID1          | Material identification number for the membrane. (Integer $\geq 0$ or blank)  |
| T             | Default membrane thickness for $T_i$ on the connection entry. If T is blank then the thickness must be specified for $T_i$ on the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 entries. (Real or blank) Average thickness if TFLAG = 1 ( $T_i = T_i \cdot T$ ) .    |
| MID2          | Material identification number for bending. (Integer $\geq -1$ or blank)  |
| $12I/T^{**3}$ | Bending moment of inertia ratio, $12I/T^3$ . Ratio of the actual bending moment inertia of the shell, $I$ , to the bending moment of inertia of a homogeneous shell, $T^3/12$ . The default value is for a homogeneous shell. (Real > 0.0; Default = 1.0) |
| MID3          | Material identification number for transverse shear. If MID2 is blank or -1, then MID3 must be blank. (Integer > 0 or blank)  |
| TS/T          | Transverse shear thickness ratio, $T_s/T$ . Ratio of the shear thickness, ( $T_s$ ), to the membrane thickness of the shell, $T$ . The default value is for a homogeneous shell. (Real > 0.0; Default = .833333)  |
| NSM           | Nonstructural mass per unit area. (Real)  |
| Z1, Z2        | Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. See Remark 11. for defaults. (Real or blank)                             |
| MID4          | Material identification number for membrane-bending coupling. See Remarks. (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2.)  |

**Remarks:**

1. PSHELL is a primary property entry. Primary properties are used to internally group entities. Therefore it is highly recommended that PSHELL entries use unique identification numbers (PIDs) with respect to all other property entries to avoid unexpected grouping results. PSHELL, PCOMP, and PCOMPG entries must have unique PIDs.
2. The structural mass is calculated from the density using the membrane thickness and membrane material properties. If MID1 is blank, then the density is obtained from the MID2 material.
3. The results of leaving an MIDi field blank (or MID2 = -1) are:

|         |   |
|---------|---|
| MID1    | No membrane or coupling stiffness   |
| MID2    | No bending, coupling, or transverse shear stiffness   |
| MID3    | No transverse shear flexibility   |
| MID4    | No bending-membrane coupling unless ZOFFS is specified on the connection entry. See Remark 6. |
| MID2=-1 | See Remark 12.  |

**Note:** **MID1 and MID2 must be specified if the ZOFFS field is also specified on the connection entry.**

4. The continuation entry is not required.
5. The structural damping (GE on the MATi entry) is obtained from the MID1 material. If MID1 is blank, then it is obtained from the MID2 material. If PARAM,SHLDAMP,DIFF is set or is anything other than SAME, then the structural damping  $K^4$  matrix is computed using the GE entries on the MATi entries according to rules in the following table (Table 32). If a single PSHELL corresponds to row eight (8) of Table 32, then all PSHELLs in the model will follow the rule of row eight (8). Rows 1-7 is an attempt to maintain upward compatibility, if a user inadvertently places a SHLDAMP,DIFF in the model.

**Note:** Large values of damping associated with an MID4 entry, when using PARAM,SHLDAMP,DIFF, can cause structural instability in transient dynamics.

Table 32 SHELL Structural Damping Rules

| SHELL Structural Damping Rules   |      |      |      |      |   |
|--|------|------|------|------|---|
| Row  | MID1 | MID2 | MID3 | MID4 | K <sup>4</sup> based on   |
| 1*   | v    | v    |      |      |   |
| 2  | v    |      |      |      | GE on MID1  |
| 3  | v    | -1   |      |      |   |
| 4  | v    | v    |      |      |   |
| 5  |      | v    |      |      | GE on MID2  |
| 6  |      | v    | v    |      |   |
| 7  | v1   | v2   | v3   | v4   | GE on MID1 if<br><br>$n = m$ and $ge_1 = ge_2 = \dots = ge_m$ or<br>$m = 1$ and $ge_1 \neq 0$ or<br>$m = 0$<br><br>where:<br>$n$ is total number of non blank MIDi (vi)<br>$m$ is total number of non zero $ge_i$ |
| 8  | v1   | v2   | v3   | v4   | Otherwise:<br><br>$GE = ge_1 \cdot \text{membrane-stiff} +$<br>$ge_2 \cdot \text{bending-stiff} +$<br>$ge_3 \cdot \text{transverse shear-stiff} +$<br>$ge_4 \cdot \text{bending-membrane-stiff}$                  |
| $* v \rightarrow \text{MIDi values the same}; vi \rightarrow \text{MIDi values different or blank}$<br>$ge_i \rightarrow \text{GE value from a MATj entry associated with MIDi}$ |      |      |      |      |   |
| If for row eight (8), a $ge_i = 0$ , it is replaced by $ge_i = 1.0E-8$   |      |      |      |      |   |

### PSHELL rules with extended MAT2 entries

The PSHELL has four MIDi entries: MID1-Membrane; MID2-Bending; MID3-Transverse Shear; MID4-Membran-Bending Coupling.

The Damping for shell elements is determined by the GE value associated to the material entry specified by MID1 (or MID2 if MID1 is blank).

PSHELL allows any of its MIDi fields to use any combination of MAT1, MAT2, or MAT8 material entries.

The MAT2 entry has a feature consisting of six (6) structural damping coefficients called GE11, GE12, GE13, GE22, GE23, and GE33 in a *second continuation entry*. When any one or more of these GEij entries are non-zero, the GE entry in field six (6) of the first continuation entry is ignored for that material.

### PSHELL rules with extended MAT2 entries

MAT2 entries with any GEij are called *extended* MAT2 entries. The damping is computed for such a material as  $G_{ij} = G_{ij} * GE_{ij}$  where  $G_{ij}$  are the elastic material coefficients on the parent MAT2 entry. An element structural damping matrix is then generated using these scaled  $G_{ij}$ .

Any PSHELL with an *extended* MAT2 in the MID1 field assumes that every MID2 and MID4 present will perform a  $G_{ij} = G_{ij} * GE_{ij}$  calculation to form the appropriate structural damping matrix. The MID3 calculation is of the form  $J_{11} = J_{11} * GE_{11}$ ,  $J_{12} = J_{12} * GE_{12}$ , and  $J_{22} = J_{22} * GE_{22}$ .

|   |   |
|---|---|
| 1 | For the case where MID1 is <i>extended</i> (is a reference to a MAT2 with GEij defined) but MID2, MID3, and/or MID4 contain a MAT1, a non- <i>extended</i> MAT2, or a MAT8 entry the calculation is of the form $G_{ij} = G_{ij} * GE$ where GE is the structural damping coefficient of the associated MAT1, MAT2, or MAT8 entry.  |
| 2 | For the case where MID1 is a MAT1, a non- <i>extended</i> MAT2, or a MAT8, but any one of the MID2, MID3, or MID4 fields have <i>extended</i> MAT2 entries, the structural damping coefficient GE of the MATi entry is used to simply scale the element stiffness as K4 matrix contribution ? GE * K.   |
| 3 | For case two (2) above, the user has the option of using PARAM,SHLDAMP,DIFF in which case any MID2, MID3, or MID4 containing an <i>extended</i> MAT2 entry force the use of the extended fields. HOWEVER, remember the PARAM,SHLDAMP,DIFF is global – meaning EVERY PSHELL scales each of its MIDI material by the appropriate GEij or GE to form an element structural damping matrix. |

6. The following should be considered when using MID4.
  - The MID4 field should be left blank if the material properties are symmetric with respect to the middle surface of the shell. If the element centerline is offset from the plane of the grid points but the material properties are symmetric, the preferred method for modeling the offset is by use of the ZOFFS field on the connection entry. Although the MID4 field may be used for this purpose, it may produce ill-conditioned stiffness matrices (negative terms on factor diagonal) if done incorrectly.
  - Only one of the options MID4 or ZOFFS should be used; if both methods are specified the effects are cumulative. Since this is probably not what the user intends, this may result in unexpected answers. Note that the mass properties are not modified to reflect the existence of the offset when the ZOFFS and MID4 methods are used. If the weight or mass properties of an offset plate are to be used in an analysis, the RBAR method must be used to represent the offset. See [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*.
  - The effects of MID4 are not considered in the calculation of differential stiffness. Therefore, it is recommended that MID4 be left blank in buckling analysis.
7. This entry is referenced by the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries via PID.
8. For structural problems, MIDI must reference a MAT1, MAT2, or MAT8 material property entry

9. If the transverse shear material MID3 or the membrane-bending coupling term MID4 references a MAT2 entry, then G33 must be zero. If MID3 references a MAT8 entry, then G1Z and G2Z must not be zero.
10. For heat transfer problems, MIDi must reference a MAT4 or MAT5 material property entry.
11. The default for Z1 is -T/2, and for Z2 is +T/2. T is the local plate thickness defined either by T on this entry or by membrane thicknesses at connected grid points, if they are input on connection entries.
12. For plane strain analysis, set MID2=-1 and set MID1 to reference a MAT1 entry. In-plane loads applied to plain strain elements are interpreted as line-loads with a value equal to the load divided by the thickness. Thus, if a thickness of "1.0" is used, the value of the line-load equals the load value. Pressure can be approximated with multiple line loads where the pressure value equals the line-load divided by the length between the loads. For SOL 600, plain strain models must be in the basic coordinate system X-Y plane.
13. For a material nonlinear property, MID1 must reference a MATS1 entry and be the same as MID2, unless a plane strain (MID2 = -1) formulation is desired. Also, MID3 cannot reference a MATS1 entry.
14. If transverse shear flexibility is specified for a model with curved shells where the loading is dominated by twist and shell normals are turned off (e.g., PARAM,SNORM,-1), then results may be inaccurate and may diverge when the mesh is refined. PARAM,SNORM should be set for this unique model condition.
15. For 3D contact analysis, MID2 must be non-zero and for 2D contact analysis, MID1 must be non-zero.

**PSHELL1**

## Properties of shell elements for SOL 700 only

Defines the properties of shell elements with variable shell thickness for SOL 700 only.

**Format:**

| 1       | 2   | 3   | 4    | 5    | 6       | 7      | 8   | 9       | 10 |
|---------|-----|-----|------|------|---------|--------|-----|---------|----|
| PSHELL1 | PID | MID | FORM | QUAD | NUMB    | SHFACT | REF | SPINCOR |    |
| +       | T1  | T2  | T3   | T4   | TRANSHR | SHRLCK |     |         |    |

**Example:**

|         |      |      |      |       |   |     |  |  |  |
|---------|------|------|------|-------|---|-----|--|--|--|
| PSHELL1 | 7    | 2    | BLT  | GAUSS | 5 | 0.9 |  |  |  |
|         | 10.0 | 10.0 | 10.0 | 10.0  |   |     |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Unique property number. (Integer; Required)  |
| MID        | Material number. See Remark 2. (Integer; Required)   |
| FORM       | Shell formulation. See Remark 2. (Character; Required)   |
|            | HUGHES Hughes-Liu  |
|            | BLT Belytschko-Lin-Tsay  |
|            | KEYHOFF Key-Hoff   |
|            | C0-TRIA C0 triangle  |
|            | MEMB Membrane element (no bending)   |
|            | DUMMY Dummy element (only applicable for FSI coupling)   |
| QUAD       | Type of quadrature. (Character; Default = GAUSS)   |
|            | GAUSS Gauss quadrature   |
|            | LOBATTO Lobatto quadrature   |
| NUMB       | The number of integration points through the thickness. (Integer; Default = 3 for quadrature). 1 for membrane element. Maximum value is 5. |
| SHFACT     | Shear factor. (Real; Default = 0.83333)  |
| REF        | Reference surface. (character, default=MID)  |
|            | TOP Reference surface is the top surface.  |
|            | MID Reference surface is the central surface.  |
|            | BOT Reference surface is the bottom surface.   |
| SPINCOR    | Spin correction. See Remark 5. (Character; default=NO)   |
|            | NO No SPINCOR applied  |

| Descriptor | Meaning   |
|------------|---|
| YES        | SPINCOR is applied  |
| T1 to T4   | Element thickness at the grid points. See Remark 4. (Real; Default = 0.0) |
| TRANSHR    | Method of transverse-shear calculation. (Character; See Remark 6.)        |
| LINEAR     | Linear transverse shear   |
| CONSTANT   | Constant transverse shear   |
| CONAPX     | Approximated constant transverse shear                                    |
| SHRLCK     | Shear-lock avoidance. (Character; See Remark 6.)                          |
| AVOID      | Avoid shear lockup  |
| NOAVOID    | No avoid  |

### Remarks:

- For constant thickness shell element with three-point Gauss integration, PSHELL entry is recommended.
- For CQUAD4 elements, the default of FORM for formulation is KEYHOFF.  
For CTRIA3 elements, the default of FORM for formulation is CO-TRIA.
- Property identification number (PID) must be unique.
- If the thickness T1 is set to blank or 0.0, the thickness of the shell must be defined on the CTRIA3 and CQUAD4 entry.
- The options for SPINCOR are:

- NO            No SPINCOR correction is applied.  
 YES          SPINCOR correction is applied.

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

- The following defaults apply:

|         | BLT           | HUGHES        | KEYHOFF |
|---------|---------------|---------------|---------|
| TRANSHR | Not Available | Not Available | LINEAR  |
| SHRLCK  | NOAVOID       | Not Available | AVOID   |

**PSHLN1**

## Nonlinear Property Extensions for a PSHELL or PCOMP(G) Entry

Specifies additional nonlinear properties for shell elements that point to a PSHELL or PCOMP(G) entry in SOL 400.

**Format:**

| 1      | 2    | 3    | 4    | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|------|------|------|-------|-------|---|---|---|----|
| PSHLN1 | PID  | MID1 | MID2 |       | ANAL  |   |   |   |    |
|        | “C3” | BEH3 | INT3 | BEH3H | INT3H |   |   |   |    |
|        | “C4” | BEH4 | INT4 | BEH4H | INT4H |   |   |   |    |
|        | “C6” | BEH6 | INT6 | BEH6H | INT6H |   |   |   |    |
|        | “C8” | BEH8 | INT8 | BEH8H | INT8H |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PSHLN1 | 22 | 98 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number of an existing PSHELL or PCOMP(G) entry. (Integer > 0)  |
| MID1       | Membrane material identification number. See Remark 17. (Integer $\geq 0$ or blank)  |
| MID2       | Bending material identification number. See Remark 17. (Integer $\geq 0$ or blank)   |
| ANAL       | Analysis type. ANAL = ‘IS’ - Implicit structural elements are being referred to.<br>ANAL = ‘IH’ - Implicit heat analysis elements are being referred to. ANAL = ‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH) |
| C3         | Keyword indicating that two items following apply to elements with three corner grids. (Character)   |
| C4         | Keyword indicating that two items following apply to elements with four corner grids. (Character)  |
| C6         | Keyword indicating that items following apply to elements with three corner grids and three midside grids. (Character)   |
| C8         | Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)   |
| BEHi       | Element structural behavior. See Remark 12. (Character Default: DCTN for BEH3, DCT for BEH4, and DCT for BEH8, MB for BEH6)  |
| INTi       | Integration scheme. See Remarks 11. and 13. (Character Default: LDK for INT3, L for INT4, QRI for INT8, Q for INT6)  |

| Descriptor | Meaning  |
|------------|--|
| BEHiH      | Element heat behavior. See Remark 12. (Character Default: DCT for BEH3H, BEH4H, and BEH8H, MB for BEH6H)         |
| INTiH      | Integration scheme. See Remarks 11. and 13. (Character Default: L for INT3H, L for INT4H, Q for INT8H and INT6H) |

**Remarks:**

1. The PID must point to an existing PSHELL, PCOMP, or PCOMPG Bulk Data entry and is honored only in SOL 400.
2. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
3. MID1 if blank (or 0) use the MID1 value on the PSHELL. If > 0 it will override the MID1 value on the PSHELL. MID1 is ignored for PCOMP/PCOMPG.
4. MID2:
  - a. If BEHi = DCT or DCTN: If blank (or 0) use the MID2 value on the PSHELL. If > 0 it will override the MID2 value on the PSHELL. If MID2 = -1 on the PSHELL entry it must be replaced with a positive value MID2 entry or the PSHELL should be replaced with a PLPLANE entry and a PSHLN2 entry should be used instead of a PSHLN1 entry.
  - b. If BEHi = MB: MID2 on both the PSHELL and PSHLN1 entries are ignored.
 MID2 is ignored for PCOMP/PCOMPG.
5. The MID1 or MID2 entries, were applicable, may point to MAT1, MAT2, MAT8, MATHE and MATSMA entries. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID1 or MID2 entries.  
 Caution: The primary MATHE and the secondary MATVE, MATVP, MATEP, MATF, and MATS8 entries are only associated with a Shell (CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, or CTRIA6) element if the Shell element refers to a PSHLN1 entry.  
 The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

| Implicit Structural Materials |         |         |       |
|-------------------------------|---------|---------|-------|
| MAT1                          | MAT2    | MAT8    | MATHE |
| MATVE                         | <MATVE> | <MATVE> | MATVE |
| MATVP                         | MATVP   |         |       |
| MATEP                         | MATEP   | MATEP   |       |
| MATF                          | MATF    | MATF    |       |
| MATS1                         |         | MATS8   |       |

<MATVE> refers to the ALTERNATE format for type ORTHO

6. If MID3 is not specified on the PSHELL, BEH4 = DCTN with INT4 = LDK should be used and any CQUAD8 elements using this PSHELL should have a new PSHELL with MID3 specified. If not a fatal message will be issued. MID3 is not used for PCOMP/PCOMPG.
7. If MID2 is not specified on the PSHELL or overridden with a nonzero value on the PSHLN1, BEH4 = MB. and BEH8 = MB. If not, a fatal message will be issued.

### Heat Materials

MAT4

MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

8. NSM is not currently supported when this entry is used.
9. When this entry points to a PCOMP or a PCOMPG, special restrictions should be noted for some of the PCOMP/PCOMPG entries:
  - NSM, FT and GE are not supported. If a failure theory for any ply defined on the PCOMP/PCOMG entry is required, a MATF entry should be specified for the associated material.
  - The allowable inter-laminar bond strength SB is supported. When used in conjunction with the parabolic shear option (NLMOPTS, TSHEAR), the provided value of SB is used to calculate the ply shear bond index for thick shells using the relation - Bond index = max(inter-laminar shear stress)/SB.
  - LAM=BLANK and LAM=SYM (only for PCOMP) are supported. No smearing i.e., conversion of PCOMP/PCOMG into equivalent PSHELL is supported. Conventional integration through the thickness and across all layers is used. LAM=MEM, BEND, SMEAR and SMCORE are treated in a manner similar to LAM=BLANK, i.e., the sequence of the plies and the data given for each ply is used to carry out the conventional thickness integration.
  - SOUTi is not supported for individual plies. If STRESS output is requested for a particular shell element, then integration point stresses and elastic strains in the material coordinate system for all plies of the element are printed. If the parabolic shear option is used, then integration point values of the inter-laminar stresses and the bond index are also printed.
  - Layer Composite (PCOM/PCOMG) is not supported to CTRAI6 by PSHLN1.
10. If BEHi = MB is selected on PSHLN1, any ZOFF entry on the element connection entry will be ignored with a user warning.

11. In the following table, BEHi refers to the nonlinear structural behavior of shell elements. An underlined item delineates a default.

| Implicit Structural Classification of Elements |           |                  |              |         |
|--|-----------|------------------|--------------|---------|
| Element Structural Type                        | BEHi CODE | Integration Code | Element Type | # Nodes |
| Doubly-curved thick shell                      | DCT       | L                | QUAD         | 4       |
|  |           | <u>QRI</u>       | QUAD         | 8       |
|  |           | LRIH             | QUAD         | 4       |
| Doubly-curved thin shell                       | DCTN      | <u>LDK</u>       | TRIA         | 3       |
|  |           | LDK              | QUAD         | 4       |
| Membrane three-dimensional                     | MB        | L                | QUAD         | 4       |
|  |           | Q                | QUAD         | 8       |
|  |           | L                | TRIA         | 3       |
|  |           | Q                | TRIA         | 6       |

12. In the following table, BEHiH refers to the nonlinear heat behavior of shell elements. An underlined item delineates a default.

| Heat Classification of Elements |            |                  |              |         |
|---------------------------------|------------|------------------|--------------|---------|
| Element Heat Type               | BEHiH CODE | Integration Code | Element Type | # Nodes |
| Doubly-curved thick shell       | DCT        | L                | QUAD         | 4       |
|                                 |            | Q                | QUAD         | 8       |
|                                 |            | L                | TRIA         | 3       |
| Membrane three-dimensional      | MB         | L                | QUAD         | 4       |
|                                 |            | Q                | QUAD         | 8       |
|                                 |            | L                | TRIA         | 3       |
|                                 |            | Q                | TRIA         | 6       |

13. Integration codes in Remark 11. are:

| INT CODE | Integration Type  |
|----------|---|
| L        | Linear  |
| LRIH     | Linear Reduced Integration Hourglass control (assumed strain) |
| Q        | Quadratic   |
| QRI      | Quadratic Reduced Integration                                 |
| LDK      | Linear Discrete Kirchhoff                                     |

14. For different MID cards in PSHELL card, advanced elements follow smeared formulation and only section forces and section moments output.
15. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation if BEH=PSTRS.
16. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available if BEH=PSTRS.
17. Structural damping is supported for elements which reference PSHLN1. The structural damping coefficient, GE, is supported for elements using the MAT1 entries.

**PSHLN2**

## Nonlinear Property Extensions for a PLPLANE Entry

Specifies additional nonlinear properties for plane strain, plane stress, or axisymmetric elements that point to a PLPLANE entry in SOL 400.

**Format:**

| 1      | 2    | 3    | 4      | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|------|------|--------|-------|-------|---|---|---|----|
| PSHLN2 | PID  | MID  | DIRECT | T     | ANAL  |   |   |   |    |
|        | "C3" | BEH3 | INT3   | BEH3H | INT3H |   |   |   |    |
|        | "C4" | BEH4 | INT4   | BEH4H | INT4H |   |   |   |    |
|        | "C6" | BEH6 | INT6   | BEH6H | INT6H |   |   |   |    |
|        | "C8" | BEH8 | INT8   | BEH8H | INT8H |   |   |   |    |

**Example:**

|        |    |    |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|
| PSHLN2 | 22 | 98 |  |  |  |  |  |  |  |
|--------|----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number of an existing PLPLANE entry. (Integer > 0)   |
| MID        | Material identification number. Remarks 13. (Integer > 0)  |
| DIRECT     | The layer direction for BEHi=COMPS or AXCOMP. See Remark 6. (Integer = 1 or 2; Default = 1)  |
| T          | A thickness. (Real; Default = 1.0)   |
| ANAL       | Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to.<br>ANAL = 'IH' - Implicit heat analysis elements are being referred to. ANAL = 'ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH) |
| C3         | Keyword indicating that two items following apply to elements with three corner grids. (Character)   |
| C4         | Keyword indicating that two items following apply to elements with four corner grids. (Character)  |
| C6         | Keyword indicating that two items following apply to elements with three corner grids and three midside grids. (Character)   |
| C8         | Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)   |
| BEHi       | Element structural behavior. See Remark 7. (Character Default: PLSTRN for BEH3, BEH4, BEH6, and BEH8)  |
| INTi       | Integration scheme. See Remarks 7. and 10. (Character Default: L for INT3, INT4, Q for INT6 and INT8)  |

| Descriptor | Meaning   |
|------------|---|
| BEHiH      | Element heat behavior. See Remark 9. (Character Default: PLSTRN for BEH3H, BEH4H, BEH6H, and BEH8H.)            |
| INTiH      | Integration scheme. See Remarks 9. and 10. (Character Default: L for INT3H, L for INT4H, Q for INT6H and INT8H) |

**Remarks:**

1. The PID must point to an existing PLPLANE Bulk Data entry and is honored only in SOL 400. Since these are additional nonlinear properties to the PLPLANE, the PLPLANE must still have an associated MATHP.
2. It is REQUIRED to override the MID value on the PLPLANE entry.
3. The element must lie in the x-y plane of the basic system. The CID field of the PLPLANE entry is not valid for this entry.
4. The MID entry may point to MAT1, MAT3, MATHORT, MATHE, MATG or MATSMA entry and MUST be used to override the MID field on a PLPLANE entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MAT1, MAT3, MAT8, MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, and MATSORT entries are only associated with a 2D or axisymmetric (CQUAD4, CQUAD, CQUAD8, CTRIA3, or CTRIA6) or (CTRIAX or CQUADX) element if the element refers to a PSHLN2 entry.

| Implicit Structural Materials |         |         |         |         |       |      |       |
|-------------------------------|---------|---------|---------|---------|-------|------|-------|
| MAT1                          | MAT2    | MAT3    | MAT8    | MATORT  | MATHE | MATG | MATHP |
| MATVE                         | <MATVE> | <MATVE> | <MATVE> | <MATVE> | MATVE |      |       |
| MATVP                         | MATVP   | MATVP   |         | MATVP   |       |      |       |
| MATEP                         | MATEP   | MATEP   | MATEP   | MATEP   |       |      |       |
| MATF                          | MATF    | MATF    | MATF    | MATF    |       |      |       |
| MATS1                         |         | MATS3   | MATS8   | MATSORT |       |      |       |

<MATVE> refers to the ALTERNATE format for type ORTHO  
MAT1 applicable to all BEHi codes of 7 below, except COMPS and AXCOMP.

MAT3 axisymmetric orthotropic applicable only to BEHi=AXSOLID code of 7 below.

MAT8 orthotropic applicable only to BEHi=PSTRS code of 7 below.

MATG for BEH4=COMPS or AXCOMP, INT4=L only.  
MATG has an associated field IDMEM that points to a MAT1.

With MATHE, MATHP, and MAT1 (with MATS1 or MATEP) BEH3=IPS or IAX with INT3=L is recommended.

**Heat Materials**

MAT4    MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries. The CTRIA6 remains a valid nonlinear heat transfer element BUT cannot be used in conjunction with this entry because it lies in an x-z plane and not an x-y plane.

5. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
6. The following table describes layer orientation for BEHi=COMPS or AXCOMP.

| Layer Orientation |                      |                               |         |
|-------------------|----------------------|-------------------------------|---------|
| DIRECT            | Normal to Layer edge | Layers run parallel from edge | to edge |
| 1                 | Element Y direction  | G1-G2                         | G4-G3   |
| 2                 | Element X direction  | G1-G4                         | G2-G3   |

7. In the following table, BEHi refers to the nonlinear structural behavior of 2D-solid elements. An underlined item delineates a default.

| Structural Classification of Elements   |           |  |  |                                      |
|---|-----------|--|--|--------------------------------------|
| Element Structural Type   | BEHi CODE | Integration Code                                     | Element Type   | # Nodes                              |
| Plane Stress  | PSTRS     | L<br><u>Q</u><br>QRI<br>LRIH<br>Q                    | QUAD<br>QUAD<br>QUAD<br>QUAD<br>TRIA                         | 4<br>8<br>8<br>4<br>6                |
| Plane Strain  | PLSTRN    | <u>L</u><br><u>L</u><br><u>Q</u><br>QRI<br>LRIH<br>Q | QUAD<br>TRIA<br>QUAD<br>QUAD<br>QUAD<br>TRIA                 | 4<br>3<br>8<br>8<br>4<br>6           |
| Plane Strain composite  | COMPS     | L  | QUAD   | 4                                    |
| Axisymmetric Solid  | AXSOLID   | L<br>L<br>LT<br>Q<br>QRI<br>QT<br>LRIH<br>Q          | QUAD<br>TRIA<br>QUAD<br>QUAD<br>QUAD<br>QUAD<br>QUAD<br>TRIA | 4<br>3<br>4<br>8<br>8<br>8<br>4<br>6 |
| Axisymmetric Composite  | AXCOMP    | L  | QUAD   | 4                                    |
| Incompressible Plane Strain   | IPS       | L  | TRIA   | 3                                    |
| Incompressible Axisymmetric   | IAX       | L  | TRIA   | 3                                    |
| Only BEH4 = COMPS or AXCOMP with INT4 = L may use the MATG, additionally they should not use a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE as they will suffer from hourgassing. |           |  |  |                                      |

8. Note for this entry with a BEHi=COMPS or AXCOMP, the THETA/MCID value on the element connection entry will be ignored.
9. In the table below, BEHiH refers to the nonlinear heat behavior of 2D-solid elements. An underlined item delineates a default.

| Heat Classification of Elements |            |                  |                              |                  |
|---------------------------------|------------|------------------|------------------------------|------------------|
| Element Heat Type               | BEHiH CODE | Integration Code | Element Type                 | # Nodes          |
| Planar Solid                    | PLSTRN     | L<br>L<br>Q<br>Q | QUAD<br>TRIA<br>QUAD<br>TRIA | 4<br>3<br>8<br>6 |
| Axisymmetric Solid              | AXSOLID    | L<br>L<br>Q<br>Q | QUAD<br>TRIA<br>QUAD<br>TRIA | 4<br>3<br>8<br>6 |

10. Integration codes in Remarks 7. and 9. are:

| INT CODE | Integration Type  |
|----------|---|
| L        | Linear  |
| LRIH     | Linear Reduced Integration Hourglass Control (assumed strain) |
| Q        | Quadratic   |
| QRI      | Quadratic Reduced Integration                                 |
| QT       | Quadratic with Twist  |
| LT       | Linear with Twist   |

11. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
12. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
13. The structural element damping coefficient, GE, is not supported on elements which reference PSHLN2.

**PSLDN1**

## Nonlinear Property Extensions for a PSOLID Entry

Specifies additional nonlinear properties for solid elements that point to a PSOLID entry in SOL 400.

**Format:**

| 1      | 2     | 3     | 4      | 5      | 6      | 7 | 8 | 9 | 10 |
|--------|-------|-------|--------|--------|--------|---|---|---|----|
| PSLDN1 | PID   | MID   | DIRECT |        | ANAL   |   |   |   |    |
|        | “C4”  | BEH4  | INT4   | BEH4H  | INT4H  |   |   |   |    |
|        | “C5”  | BEH5  | INT5   | BEH5H  | INT5H  |   |   |   |    |
|        | “C6”  | BEH6  | INT6   | BEH6H  | INT6H  |   |   |   |    |
|        | “C8”  | BEH8  | INT8   | BEH8H  | INT8H  |   |   |   |    |
|        | “C10” | BEH10 | INT10  | BEH10H | INT10H |   |   |   |    |
|        | “C13” | BEH13 | INT13  | BEH13H | INT13H |   |   |   |    |
|        | “C15” | BEH15 | INT15  | BEH15H | INT15H |   |   |   |    |
|        | “C20” | BEH20 | INT20  | BEH20H | INT20H |   |   |   |    |

**Example:**

|        |    |    |   |  |  |  |  |  |  |
|--------|----|----|---|--|--|--|--|--|--|
| PSLDN1 | 22 | 55 | 2 |  |  |  |  |  |  |
|--------|----|----|---|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| PID        | Property identification number of an existing PSOLID entry. (Integer > 0)  |
| MID        | Material identification number. Remark 8.(Integer $\geq 0$ )   |
| DIRECT     | The layer direction for BEHi=SLCOMP. See Remark 5. (Integer, 1, 2, OR 3; Default = 1)  |
| ANAL       | Analysis type. ANAL = ‘IS’ - Implicit structural elements are being referred to.<br>ANAL = ‘IH’ - Implicit heat analysis elements are being referred to. ANAL = ‘ISH’ - Implicit structural and heat elements are being referred to. (Character Default ISH) |
| C4         | Keyword indicating that two items following apply to elements with four corner grids. (Character)  |
| C5         | Keyword indicating that two items following apply to elements with five corner grids. (Character)  |
| C8         | Keyword indicating that two items following apply to elements with eight corner grids. (Character)   |
| C10        | Keyword indicating that two items following apply to elements with four corner grids and six midside grids. (Character)  |
| C13        | Keyword indicating that two items following apply to elements with five corner grids and eight midside grids. (Character)  |
| C15        | Keyword indicating that items following apply to elements with six corner and nine midside grids. (Character)  |

| Descriptor | Meaning   |
|------------|---|
| C20        | Keyword indicating that two items following apply to elements with eight corner grids and twelve midside grids. (Character) |
| BEHi       | Element structural behavior. See Remark 6. (Character default: SOLID for BEH4, BEH6, BEH8, BEH10, BEH15, and BEH20)         |
| INTi       | Integration scheme. See Remark 7. (Character default: L for INT4, INT6, and INT8; Q for INT10, INT15, and INT20)            |
| BEHiH      | Element heat behavior. See Remark 6. (Character Default: SOLID for BEH4H, BEH6H, BEH8H, BEH10H, BEH15, and BEH20H)          |
| INTiH      | Integration scheme. See Remark 7. (Character Default: L for INT4H, INT6H, and INT8H; Q for INT10H, INT15H, and INT20H)      |

### Remarks:

1. The PID must point to an existing PSOLID Bulk Data entry and is honored only in SOL 400.
2. The MID entry may point to MAT1, MAT9, MATORT, MATHE, or MATG entries and can be used to override the MID field on a PSOLID entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP and MATF entries are only associated with a 3D Solid (CHEXA, CPENTA, CPYRAM and CTETRA) element if the Solid element refers to a PSLDN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

| Implicit Structure Materials |         |         |       |      |       |
|------------------------------|---------|---------|-------|------|-------|
| MAT1                         | MAT9    | MATORT  | MATHE | MATG | MATHP |
| MATVE                        | <MATVE> | <MATVE> | MATVE |      |       |
| MATVP                        | MATVP   | MATVP   |       |      |       |
| MATEP                        | MATEP   | MATEP   |       |      |       |
| MATF                         | MATF    | MATF    |       |      |       |
| MATS1                        |         | MATSORT |       |      |       |

<MATVE> refers to the ALTERNATE format for type ORTHO  
MATG for BEH8=SLCOMP, INT8=L only.

MATG has an associated field IDMEM that points to a MAT1  
BEH4 = ISOL, INT4 = L is currently limited to MAT1 (with possible association with MATS1 or  
MATEP), MATHE and MATHP

### Heat Materials

MAT4      MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

3. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
4. The following table describes layer orientation for BEH8=SLCOMP, INT8=L.

| Layer Orientation |                       |                               |             |
|-------------------|-----------------------|-------------------------------|-------------|
| DIRECT            | Normal to Layer plane | Layers run parallel from face | to face     |
| 1                 | Element T direction   | G1-G2-G3-G4                   | G5-G6-G7-G8 |
| 2                 | Element R direction   | G1-G4-G8-G5                   | G2-G3-G7-G6 |
| 3                 | Element S direction   | G1-G2-G6-G5                   | G4-G3-G7-G8 |

5. In the following table, BEHi refers to the nonlinear structural behavior of the solid element. An underlined item delineates a default.

| Structural Classification of Elements |           |                  |              |         |
|---------------------------------------|-----------|------------------|--------------|---------|
| Element Structural Type               | BEHi CODE | Integration Code | Element Type | # Nodes |
| SOLID                                 | SOLID     | L                | HEX          | 8       |
|                                       |           | <u>Q</u>         | HEX          | 20      |
|                                       |           | QRI              | HEX          | 20      |
|                                       |           | LRIH             | HEX          | 8       |
|                                       |           | <u>Q</u>         | TET          | 10      |
|                                       |           | LRIH             | TET          | 10      |
|                                       |           | <u>L</u>         | TET          | 4       |
|                                       |           | <u>L</u>         | PEN          | 6       |
|                                       |           | <u>Q</u>         | PEN          | 15      |
|                                       |           | <u>L</u>         | PYR          | 5       |
| Solid continuum composite             | SLCOMP    | <u>L</u>         | PYR          | 5       |
|                                       |           | ASTN*            | PYR          | 13      |

| Structural Classification of Elements  |           |                  |              |         |
|--|-----------|------------------|--------------|---------|
| Element Structural Type  | BEHi CODE | Integration Code | Element Type | # Nodes |
| Solid Shell  | SLCOMP    | ASTN             | HEX          | 8       |
| Incompressible solid   | ISOL      | L                | TET          | 4       |
| Only BEH8=SLCOMP,INT=L may use a MATG, additionally it should not use a MAT1, MAT9, MATORT, or MATHE as it will suffer from hour-glassing. |           |                  |              |         |
| It is recommended that for BEH4 = ISOL, INT4 = L that NLMOPTS,LRGSTRN,2 be flagged   |           |                  |              |         |
| *Only DIRECT=1 is allowed.   |           |                  |              |         |

6. In the following table, BEHiH refers to the heat behavior of the solid element. An underlined item delineates a default.

| Heat Classification of Elements |            |                  |              |         |
|---------------------------------|------------|------------------|--------------|---------|
| Element Heat Type               | BEHiH CODE | Integration Code | Element Type | # Nodes |
| SOLID                           | SOLID      | L                | HEX          | 8       |
|                                 |            | <u>Q</u>         | HEX          | 20      |
|                                 |            | <u>Q</u>         | TET          | 10      |
|                                 |            | <u>L</u>         | TET          | 4       |
|                                 |            | <u>L</u>         | PEN          | 6       |
|                                 |            | <u>Q</u>         | PEN          | 15      |
|                                 |            | <u>L</u>         | PYR          | 5       |
|                                 |            | <u>Q</u>         | PYR          | 13      |

7. Integration codes in Remark 5. are:

| INT CODE | Integration Type  |
|----------|---|
| L        | Linear  |
| LRIH     | Linear Reduced Integration Hourglass control (assumed strain) |
| ASTN     | Assumed STraN enhanced formulation solid shell                |
| Q        | Quadratic   |
| QRI      | Quadratic Reduced Integration                                 |

8. Structural damping is supported for elements which reference PSLDN1.
- The structural damping coefficient, GE, is supported for elements using the MAT1 or MAT9 entries.
  - Extended damping coefficients GEij are supported for elements using the MAT9 entries.

**PSOLID**

## Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, CPYRAM and CTETRA entries).

**Format:**

| 1      | 2   | 3   | 4     | 5  | 6      | 7    | 8    | 9 | 10 |
|--------|-----|-----|-------|----|--------|------|------|---|----|
| PSOLID | PID | MID | CORDM | IN | STRESS | ISOP | FCTN |   |    |

**Example:**

|        |   |     |   |     |      |         |      |  |  |
|--------|---|-----|---|-----|------|---------|------|--|--|
| PSOLID | 2 | 100 | 6 | TWO | GRID | REDUCED | PORO |  |  |
|--------|---|-----|---|-----|------|---------|------|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0)   |
| MID        | Identification number of a MAT1, MAT4, MAT5, MAT9, MAT10, MATHP, MATPE1, or MATHE entry. (Integer > 0)  |
| CORDM      | Identification number of the material coordinate system. See Remarks 3. and 4. (Integer; Default = 0, which is the basic coordinate system; see Remark 3.)  |
| IN         | Integration network. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)   |
| STRESS     | Location selection for stress output. See Remarks 9. and 10. (Integer; Character, or blank)   |
| ISOP       | Integration scheme. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)  |
| FCTN       | Fluid element flag. (Character: “FFLUID” indicates a fluid element with frequency dependent rigid absorber properties, “PFLUID” indicates a fluid element, “SMECH” indicates a structural element, “PORO” indicates an element of poroelastic medium, see Remark 12.; Default = “SMECH.”) |

**Remarks:**

1. PSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS entries.
2. Isotropic (MAT1 or MAT4), anisotropic (MAT5 or MAT9), or fluid (MAT10) material properties may be referenced. If FCTN = “PFLUID” or “FFLUID”, then MID must reference a MAT10 entry. PFLUID and FFLUID are not available for SOL 600 and SOL 700.

3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate systems. The material coordinate system (CORDM) may be the basic system (0 or blank), any defined system (Integer > 0), or the standard internal coordinate system of the element based on eigenvalue techniques to insure non bias in the element formulation designated as element coordinate system (-1) or the element system (-2). The default value for CORDM is zero unless it is overridden by the NASTRAN statement with the CORDM keyword. When MAT9 entry is used, the coordinate system used effects the stiffness calculation of the element. See [Executing MSC Nastran, 1](#).
4. For the CPYRAM entry the element coordinate system is the same as the basic coordinate system. CORDM values of 0, -1, -2 or blank are treated to be identical to the basic coordinate system.
5. Advanced nonlinear solid elements identified through an additional PSLDN1 entry, or SPROPMAP > -1 in NLMOPTS, or IDAMP>0 in NLSTEP, and so on, in SOL 400 do not support: IN, ISOP, and FCTN. Also, in this case, CORDM=-1 is only supported for CHEXA elements. Fatal error is issued if CORRDM < 0 is used for CPENTA and CTETRA.

**The following Remarks, DO NOT APPLY TO SOL 600 or SOL 700.**

6. For CHEXA and CPENTA elements with no midside nodes, reduced shear integration with bubble functions (ISOP = blank or "REDUCED" and IN = blank or "BUBBLE") is the default. This is recommended because it minimizes shear locking and Poisson's ratio locking and does not cause modes of deformation that lead to no strain energy. The effects of using nondefault values are as follows:
  - a. IN = "THREE" or 3 produces an overly stiff element.
  - b. If IN = "TWO" or 2 and the element has midside nodes, modes of deformation may occur that lead to no strain energy.
  - c. Standard isoparametric integration (ISOP = "FULL" or 1 and IN = "TWO" or 2; or "THREE" or 3) produces an element overly stiff in shear. This type of integration is more suited to nonstructural problems.
  - d. In SOL105 the default reduced integration scheme often produces spurious modes in which case it is recommended that ISOP="FULL" and IN="THREE" be used.
7. IN = "BUBBLE" is not allowed for CTETRA elements or for CHEXA and CPENTA elements with midside nodes.
8. If you use IN = "BUBBLE" for CTETRA elements, NASTRAN internally switch to IN=2 if you have 4-noded CTETRA element and IN=3 greater than 4 nodes.
9. Stress output may be requested at the Gauss points (STRESS = "GAUSS" or 1) of linear CHEXA and CPENTA elements (i.e. elements with no midside nodes). Gauss point output is available for the linear and quadratic CTETRA and CPYRAM elements (i.e. elements with or without midside nodes).
10. The following tables indicate the allowed options and combination of options. If a combination not found in the table is used, then a warning message will be issued and default values will be assigned for all options.
11. The gauss point locations for the solid elements are documented in [Nonlinear Analysis, 535](#) in the *MSC Nastran Reference Guide*.

12. The entries with FCTN="PORO" must select MATPE1 identification number in MID field.

Table 9-32 CHEXA Entry Options

| CHEXA      | Integration  | IN                             | STRESS<br>(Default:<br>GRID) | ISOP (Default: See<br>Remarks 5 and 7.) | Nonlinear Capability        |            |            |  |
|------------|--|--------------------------------|------------------------------|---|-----------------------------|------------|------------|--|
|            |  |                                |                              |   | SOLs<br>106,<br>129,<br>400 | SOL<br>600 | SOL<br>700 |  |
| 8 Node     | 2x2x2 Reduced Shear with Bubble Function (default) | BUBBLE or Blank or 0 (Default) | Blank or GRID or GAUSS or 1  | Blank or REDUCED (Default*)             | Yes                         | Yes**      | Yes        |  |
|            | 2x2x2 Reduced Shear Only                           | TWO or 2                       |                              |   |                             |            |            |  |
|            | 2x2x2 Standard Isoparametric                       | FULL or 1                      |                              |   | Yes                         | No         |            |  |
|            | 3x3x3 Reduced Shear Only                           | THREE or 3                     | Blank or GRID                | Blank or REDUCED                        | No                          | Yes        | Yes        |  |
|            | 3x3x3 Standard Isoparametric                       |                                |                              | FULL or 1                               |                             | Yes**      | No         |  |
| 9-20 Node  | 2x2x2 Reduced Shear Only                           | TWO or 2                       | Blank or GRID                | Blank or REDUCED                        | No                          | Yes        | No         |  |
|            | 2x2x2 Standard Isoparametric                       |                                |                              | FULL or 1                               |                             | Yes        | No         |  |
|            | 3x3x3 Reduced Shear Only (default)                 | Blank or THREE or 3 (Default)  |                              | Blank or REDUCED (Default*)             |                             | No         | No         |  |
|            | 3 x3x3 Standard Isoparametric                      |                                |                              | FULL or 1                               |                             | Yes        | No         |  |
| p-elements | Reduced (p-order) Bubble                           | 0 or 1                         | Not applicable               | 0                                       | No                          | No         | No         |  |
|            | Bubble, P+ISOP Integration                         | 1                              |                              | -10 ≤ ISOP ≤ -10                        |                             |            |            |  |
|            | No Bubble, P+ISOP Integration                      | 2 or 3                         |                              |   |                             |            |            |  |

\*REDUCED is the default only for structural elements (FCTN="SMECH").

\*\* Requires PARAM,MRALIAS

Table 9-33 CPENTA Entry Options

| CPENTA    | Integration                                      | IN                             | STRESS<br>(Default:<br>GRID) | ISOP (Default:<br>See Remarks 5<br>and 7.) | Nonlinear Capability     |            |            |
|-----------|--|--------------------------------|------------------------------|--|--------------------------|------------|------------|
|           |  |                                |                              |  | SOLs<br>106, 129,<br>400 | SOL<br>600 | SOL<br>700 |
| 6 Node    | 2x3 Reduced Shear with Bubble Function (Default) | Blank or 0 or BUBBLE (Default) | GAUSS or 1 or Blank or GRID  | Blank or REDUCED (Default*)                | Yes                      |            |            |
|           | 2x3 Reduced Shear Only                           |                                |                              |  |                          | Yes**      | Yes        |
|           | 2x3 Standard Isoparametric                       |                                |                              | FULL or 1                                  |                          | Yes        | No         |
|           | 3x7 Reduced Shear Only                           | THREE or 3                     | Blank or GRID                | Blank or REDUCED                           | No                       | No         | No         |
|           | 3x7 Standard Isoparametric                       |                                |                              | FULL or 1                                  |                          | No         | No         |
| 7-15 Node | 2x3 Reduced Shear Only                           | TWO or 2                       | Blank or GRID                | Blank or REDUCED                           | No                       | Yes**      | No         |
|           | 2x3 Standard Isoparametric                       |                                |                              | FULL or 1                                  |                          | Yes        | No         |
|           | 3x7 Reduced Shear Only (default)                 | Blank or THREE or 3 (Default)  |                              | Blank or REDUCED (Default*)                |                          | No         | No         |
|           | 3x7 Standard Isoparametric                       |                                |                              | FULL or 1                                  |                          | No         | No         |

\*\* Requires PARAM,MRALIAS

|   |   |        |                |  |    |    |    |
|---|---|--------|----------------|--|----|----|----|
| p-elements<br>Bubble Function                                 | 2x3 for p=1, 1, 1<br>3x7 for p=2, 2, 2<br>(p+1)x(p) for all other | 0 or 1 | Not applicable | 0                                      | No |    |    |
|   | 2x3 for p=1, 1, 1<br>3x7 for p=2, 2, 2                            |        |                | 0                                      |    |    |    |
|   | 3x7 for p=1, 1, 1   |        |                | 1                                      |    |    |    |
|   | (p+ISOP+1) x (p+ISOP) for all other                               |        |                | -10 ≤ ISOP ≤ -1<br>or<br>2 ≤ ISOP ≤ 10 |    | No | No |
| p-elements<br>Standard Isoparametric;<br>(no bubble function) | 2x3 for p=1, 1, 1<br>3x7 for p=2, 2, 2                            | 2 or 3 |                | 0                                      |    |    |    |
|   | 3x7 for p=1, 1, 1   |        |                | 1                                      |    |    |    |
|   | (p+ISOP+1) x (p+ISOP) for all other                               |        |                | -10 ≤ ISOP ≤ -1<br>or<br>2 ≤ ISOP ≤ 10 |    |    |    |
|   |   |        |                |  |    |    |    |

\*REDUCED is the default only for structural elements (FCTN="SMECH").

Table 9-34 CTETRA Entry Options

| CTETRA     | Integration   | IN                            | STRESS<br>(Default:<br>GRID) | ISOP                                   | Nonlinear Capability  |            |            |
|------------|---|-------------------------------|------------------------------|--|-----------------------|------------|------------|
|            |   |                               |                              |  | SOLs 106,<br>129, 400 | SOL<br>600 | SOL<br>700 |
| 4 Node     | 1-Point Standard Isoparametric (Default)                        | Blank or TWO or 2 (Default)   | GAUSS or 1 or Blank or GRID  | Blank or FULL                          | Yes                   | Yes**      | Yes        |
|            | 5-Point Standard Isoparametric                                  | THREE or 3                    | Blank or GRID                |  | No                    | Yes        | No         |
| 5-10 Node  | 5-Point Standard Isoparametric                                  | Blank or THREE or 3 (Default) | GAUSS or 1 or Blank or GRID  | Blank or FULL                          | Yes                   | Yes        | No         |
| p-elements | 1-Point; P=1,1,1<br>5-Point; P=2,2,2<br>P+1 Cubic Point;<br>P>2 | 0 or 1                        | Not applicable               | 0                                      | No                    | No         | No         |
|            | 5-Point; P=1,1,1<br>P+1 Cubic for all other                     |                               |                              | 1                                      |                       |            |            |
|            | P+ISOP Cubic  |                               |                              | -10 ≤ ISOP ≤ -1<br>or<br>2 = ISOP ≤ 10 |                       |            |            |

\*\*Requires PARAM,MRALIAS

Table 9-35 CPYRAM Entry Options

| CPYRAM  | Integration                              | IN             | STRESS<br>(Default:<br>GRID) | ISOP           | Nonlinear Capability |            |            |
|---------|--|----------------|------------------------------|----------------|----------------------|------------|------------|
|         |  |                |                              |                | SOL 400              | SOL<br>600 | SOL<br>700 |
| 5 Node  | 5-Point Standard Isoparametric (Default) | Not applicable | GAUSS or 1 or Blank or GRID  | Not applicable | Yes                  | No         | No         |
|         | 5-Point Standard Isoparametric           |                |                              |                | Yes                  | No         | No         |
| 13 Node | 8-Point Standard Isoparametric           | Not applicable | GAUSS or 1 or Blank or GRID  | Not applicable | Yes                  | No         | No         |

**PSPRMAT**

## Spring and damper elements for translation and rotation - SOL 700

Defines linear and nonlinear spring element property for translation. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3       | 4       | 5    | 6      | 7 | 8 | 9 | 10 |
|---------|-----|---------|---------|------|--------|---|---|---|----|
| PSPRMAT | PID |         |         |      |        |   |   |   |    |
|         |     |         |         |      |        |   |   |   |    |
|         | K   | FAILMTF | FAILMCF | LOAD | UNLOAD |   |   |   |    |

**Example:**

|         |        |  |  |  |  |  |  |  |  |
|---------|--------|--|--|--|--|--|--|--|--|
| PSPRMAT | 1      |  |  |  |  |  |  |  |  |
|         |        |  |  |  |  |  |  |  |  |
|         | 20.0E3 |  |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| PID        | Property ID. PID is referenced on the CSPR entry. (Integer > 0; required.)   |
| K          | Elastic stiffness (force/displacement). (Real > 0; See Remarks 1. and 2.)  |
| FAILMTF    | Tensile failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)   |
| FAILMCF    | Compressive failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)   |
| LOAD       | Number of a TABLED1 entry defining the variation of force (y-value) with displacement (x-value) during loading. (Integer > 0; See Remarks 1., 3. and 4.) |
| UNLOAD     | Number of a TABLExx entry defining the variation of force (y-value) with displacement (x-value) during unloading. (Integer > 0; See Remarks 3. and 4.)   |

**Remarks:**

- Only one set of K, FAILMTF and FAILMCF or LOAD and UNLOAD is allowed. If K is defined, LOAD and ULOAD are not allowed. If LOAD is defined, K, FAILMTF and FAILMCF are not allowed.
- For linear spring, K is required and FAILMTF and FAILMCF are optional.
- For nonlinear spring, LOAD is required and UNLOAD is optional. If UNLOAD is not defined, unloading path is identical as LOAD.
- Input for loading and unloading must be consistent. Both curves must be either completed defined or have only positive values (start from (0.,0.). When only positive values are defined, the curves are automatically mirrored.

**PSSHL** Properties for Solid Shell (CSSHL) Elements in SOL 600

Defines the properties for Solid Shell (CSSHL) elements in SOL 600.

**Format:**

| 1     | 2   | 3   | 4  | 5  | 6    | 7 | 8 | 9 | 10 |
|-------|-----|-----|----|----|------|---|---|---|----|
| PSSHL | PID | MID | IT | SF | MCID |   |   |   |    |

**Example:**

|       |    |    |  |       |  |  |  |  |  |
|-------|----|----|--|-------|--|--|--|--|--|
| PSSHL | 11 | 33 |  | .8333 |  |  |  |  |  |
|-------|----|----|--|-------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| PID        | Property identification number. (Integer > 0; Required)   |
| MID        | Identification of a MATxxx entry. All MAT entries available in SOL 600 can be specified except for hyperelastic materials. (Integer > 0)  |
| IT         | Transition thickness - Enter only if a solid shell is attached to a standard shell (such as CQUAD4), in which case TT is the thickness of the standard shell. (Real; Default = 0.0) |
| SF         | Transverse shear factor - Leave blank if transverse shear is not to be considered. (Real or blank, if entered SF must range between 0.0 and 1.0)                                    |
| CORDM      | Identification number of a material coordinate system. See Remark 3. (Integer; Default = 0)   |

**Remarks:**

1. PSSHL entries should have unique identification numbers with respect to all other property entries.
2. MID may reference isotropic, orthotropic or anisotropic materials with or with plasticity, however hyperelastic and foam materials are not available.
3. See the CHEXA entry for the definition of the element coordinate system. The material coordinate system (CORDM) may be the basic system (0 or blank) or any defined system (Integer > 0), or the element coordinate system (-1). The default value for CORDM is zero.

**PTUBE****Tube Property**

Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).

**Format:**

| 1     | 2   | 3   | 4  | 5 | 6   | 7   | 8 | 9 | 10 |
|-------|-----|-----|----|---|-----|-----|---|---|----|
| PTUBE | PID | MID | OD | T | NSM | OD2 |   |   |    |

**Example:**

|       |   |   |      |      |  |  |  |  |  |
|-------|---|---|------|------|--|--|--|--|--|
| PTUBE | 2 | 6 | 6.29 | 0.25 |  |  |  |  |  |
|-------|---|---|------|------|--|--|--|--|--|

**Describer      Meaning**

|     |   |
|-----|---|
| PID | Property identification number. (Integer > 0)                                   |
| MID | Material identification number. See Remarks 3. and 4. (Integer > 0)             |
| OD  | Outside diameter of tube. (Real > 0.0)  |
| T   | Thickness of tube. (Real; T ≤ OD / 2.0 )  |
| NSM | Nonstructural mass per unit length. (Real)                                      |
| OD2 | Diameter of tube at second grid point (G2) on CTUBE entry. (Real; Default = OD) |

**Remarks:**

1. If T is zero, a solid circular rod is assumed.
2. PTUBE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PTUBE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PTUBE entries.
3. For structural problems, MID must reference a MAT1 material entry.
4. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
5. Tapered OD tubes with constant wall thickness are available for heat transfer only. The effective diameter is given by:

$$D_{\text{effective}} = T + \frac{D_2 - D_1}{\log_e\left(\frac{D_2 - T}{D_1 - T}\right)}$$

where:

$$D_1 = \text{OD}$$

$$D_2 = \begin{cases} \text{OD2 if } \text{OD2} \neq 0 \\ \text{OD if } \text{OD2} = 0 \text{ or blank} \end{cases}$$

**PVAL****p-Version Element Polynomial Order Distribution**

Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

**Format:**

| 1    | 2  | 3     | 4     | 5     | 6   | 7      | 8  | 9 | 10 |
|------|----|-------|-------|-------|-----|--------|----|---|----|
| PVAL | ID | POLY1 | POLY2 | POLY3 | CID | SETTYP | ID |   |    |

**Example:**

|      |     |   |   |   |  |  |  |  |  |
|------|-----|---|---|---|--|--|--|--|--|
| PVAL | 127 | 1 | 2 | 1 |  |  |  |  |  |
|------|-----|---|---|---|--|--|--|--|--|

| Descriptor | Meaning  | Type             | Default   |
|------------|--|------------------|-----------|
| ID         | ID selected in ADAPT Bulk Data entry.  | Integer > 0      | Required  |
| CID        | Coordinate system used to specify polynomial values in different directions. See Remark 1. | Integer $\geq 0$ | Remark 2. |
| POLYi      | Polynomial order in 1, 2, 3 directions of the CID system.                                  | Integer > 0      | Remark 3. |
| SETTYP     | Type of set provided (SET or ELID). See Remark 6.  | Character        | "SET"     |
| ID         | SET ID or Element ID with these p value specifications. See Remark 6.                      | Integer > 0      | 999999    |

**Remarks:**

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element's topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the mid point of this vector). The p-level of each edge is now determined by the nearest integer to:

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where  $(n_1, n_2, n_3)$  are the components of this unit vector in the CID system.

3. For accuracy and efficiency the recommended minimum p-order is 3. The default value for POLY2 and POLY3 is POLY1.

4. Each finite element has to have a unique PVAL for PSTRUPTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest  $p_i$  entry.
5. The intermediate PVAL entries generated will have an identification number starting with ADGEN.
6. Whenever SETTYP = "SET", a SET command must be defined under the SETS DEFINITION command in the Case Control Section.
7. SET = 999999 is a reserved set that includes all elements.
8. If there are more than one PVAL entries for a given element, then
  - If CID on the PVALs are the same, the entry with the maximum POLYi will be used.
  - If CID on the PVALs are different, a fatal message is issued.

**PVISC**

## Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

**Format:**

| 1     | 2    | 3   | 4   | 5 | 6    | 7   | 8   | 9 | 10 |
|-------|------|-----|-----|---|------|-----|-----|---|----|
| PVISC | PID1 | CE1 | CR1 |   | PID2 | CE2 | CR2 |   |    |

**Example:**

|       |   |     |      |  |  |  |  |  |  |
|-------|---|-----|------|--|--|--|--|--|--|
| PVISC | 3 | 6.2 | 3.94 |  |  |  |  |  |  |
|-------|---|-----|------|--|--|--|--|--|--|

**Describer      Meaning**

- PID<sub>i</sub>      Property identification number. (Integer > 0)  
CE1, CE2      Viscous damping values for extension in units of force per unit velocity. (Real)  
CR1, CR2      Viscous damping values for rotation in units of moment per unit velocity. (Real)

**Remarks:**

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.
3. PVISC is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PVISC property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PVISC entries.

**PVISC1**

## Nonlinear Damper Property- SOL 700

Defines the properties of a nonlinear damper where the damping constant varies with the velocity. Used in SOL700 only.

**Format:**

|        |     |       |   |   |   |   |   |   |    |
|--------|-----|-------|---|---|---|---|---|---|----|
| 1      | 2   | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
| PVISC1 | PID | TABLE |   |   |   |   |   |   |    |

**Example:**

|        |   |     |  |  |  |  |  |  |  |
|--------|---|-----|--|--|--|--|--|--|--|
| PVISC1 | 8 | 236 |  |  |  |  |  |  |  |
|--------|---|-----|--|--|--|--|--|--|--|

| Field | Content   |
|-------|---|
| PID   | Unique property number. (Integer > 0; required)   |
| TABLE | TABLED1 ID defining the variation of the force (y-value) with velocity (x-value). (Integer > 0; required) |

**Remarks:**

1. This entry defines the properties of a nonlinear damper. Use the PVISC entry to define linear dampers.
2. The values in the table are interpolated to get the force for a particular velocity.

**PWELD**

## Connector Element Property

Defines the property of connector (CWELD) elements.

**Format:**

| 1     | 2     | 3     | 4 | 5 | 6 | 7    | 8 | 9    | 10 |
|-------|-------|-------|---|---|---|------|---|------|----|
| PWELD | PID   | MID   | D |   |   | MSET |   | TYPE |    |
|       | LDMIN | LDMAX |   |   |   |      |   |      |    |

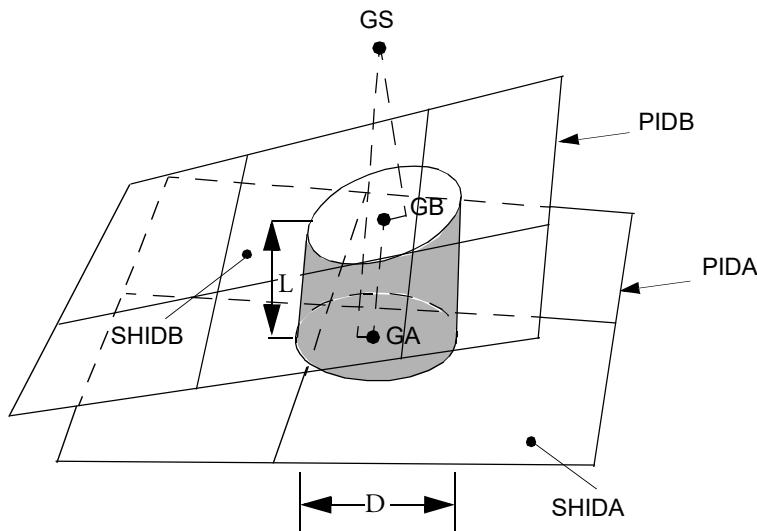
**Example:**

|       |     |   |     |  |  |  |  |  |  |
|-------|-----|---|-----|--|--|--|--|--|--|
| PWELD | 100 | 3 | 1.0 |  |  |  |  |  |  |
|-------|-----|---|-----|--|--|--|--|--|--|

| Descriptor | Meaning   | Type          | Default  |
|------------|---|---------------|----------|
| PID        | Property identification number.   | Integer > 0   | Required |
| MID        | Material identification number. See Remark 1.   | Integer > 0   | Required |
| D          | Diameter of the connector. See Remark 1.  | Real > 0      | Required |
| MSET       | Active ONLY for "PARAM,OLDWELD,YES".<br>Flag to eliminate m-set degrees-of-freedom (DOFs).<br>The MSET parameter has no effect in a nonlinear<br>SOL 400 analysis.<br><br>=OFF m-set DOFs are eliminated, constraints are<br>incorporated in the stiffness, see Remark 2.<br>=ON m-set DOFs are not eliminated, constraints<br>are generated. | Character     | OFF      |
| TYPE       | Character string indicating the type of connection,<br>see Remarks 3. and 4.<br><br>=blank general connector<br>="SPOT" spot weld connector   | Character     | Blank    |
| LDMIN      | Active ONLY for "PARAM,OLDWELD,YES".<br>Smallest ratio of length to diameter for stiffness<br>calculation, see Remark 4.  | Real or blank | 0.2      |
| LDMAX      | Active ONLY for "PARAM,OLDWELD,YES".<br>Largest ratio of length to diameter for stiffness<br>calculation, see Remark 4.   | Real or blank | 5.0      |

**Remarks:**

1. The material MID, the diameter D, and the length are used to calculate the stiffness of the connector in 6 directions. MID can only refer to the MAT1 Bulk Data entry. The length is the distance of GA to GB (see [Figure 9-150](#)).



**Figure 9-150 Length and Diameter of the CWELD Connector**

2. This remark is now valid only if "PARAM,OLDWELD,YES" is present. The parameter MSET is active only for the formats ELEMID and GRIDID (see [CWELD, 1684](#) for the format descriptions). MSET = "OFF" incorporates constraints at the element stiffness matrix level avoiding explicit m-set constraint equations. For the formats PARTPAT and ELPAT, constraints are always eliminated on the element level. MSET = "ON" generates explicit m-set constraints. For example, if a patch-to-patch connection is specified with the formats "GRIDID" or "ELEMID" on the CWELD entry, and MSET=ON is specified, 2x6 explicit constraints are generated that connect the 6 degrees-of-freedom of GA to the translational degrees-of-freedom of grid points GAi and the 6 degrees-of-freedom of GB to GBi. The 2x6 degrees-of-freedom of GA and GB are put into the m-set. The constraints are labeled "RWELD". The identification numbers of the generated RWELD constraint elements start with an offset of 100,001,001 by default. The offset number can be changed with PARAM, OSWELM. For MSET = "OFF" or blank, the 2x6 constraint equations are built into the stiffness matrix of the CWELD element, thereby condensing the 2x6 degrees-of-freedom of GA and GB.
3. TYPE = "SPOT" is good for the formats PARTPAT, ELPAT, or ELEMID with patch to patch connections. For point to patch connections or another formats of patch to patch connections, TYPE is always set to blank.

If "PARAM,OLDWELD,YES" is present, the effective length for the stiffness of the CWELD element is set to  $L_e = 1/2 \cdot (t_A + t_B)$  regardless of the distance GA to GB.  $t_A$  and  $t_B$  are the shell thicknesses of shell A and B, respectively. The effective length is used to avoid excessively stiff or soft connections due to mesh irregularities.

For the new CWELD, the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance GA to GB is equal to  $L = 1/2 \cdot (t_A + t_B)$ .

4. If TYPE=blank and "PARAM,OLDWELD,YES" is present, the effective length  $L_e$  of the CWELD is equal to the true length L, the distance of GA to GB, as long as the ratio of the length L to diameter D is in the range  $LDMIN \leq L/D \leq LDMAX$ . If L is below the range, the effective length is set to  $L_e = LDMIN \cdot D$  and if L is above the range, the effective length is set to  $L_e = LDMAX \cdot D$ .

For the new CWELD with TYPE=blank, LDMIN and LDMAX are ignored. If the distance of GA to GB (patch to patch connections) or GS to GA (point to patch connections) is less than 1.0e-4, the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance is equal to 1.0e-4.

5. PWELD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PWELD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PWELD entries.
6. The new CWELD will not contribute to MASS by default even if its associated MATi entry has a non-zero density. To react to a non-zero density "SWLDPARM,WMASS,1" is required. If mass is computed, the PARAM,COUPMASS effects the mass calculation.

## Entries Q-S

### QBDY1

Boundary Heat Flux Load for CHBDYj Elements, Form 1

Defines a uniform heat flux into CHBDYj elements.

#### Format:

| 1     | 2    | 3    | 4      | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|------|--------|------|------|------|------|------|----|
| QBDY1 | SID  | Q0   | EID1   | EID2 | EID3 | EID4 | EID5 | EID6 |    |
|       | EID7 | EID8 | -etc.- |      |      |      |      |      |    |

#### Example:

|       |     |      |     |  |  |  |  |  |  |
|-------|-----|------|-----|--|--|--|--|--|--|
| QBDY1 | 109 | 1.-5 | 721 |  |  |  |  |  |  |
|-------|-----|------|-----|--|--|--|--|--|--|

#### Alternate Format and Example:

|       |     |      |      |        |      |  |  |  |  |
|-------|-----|------|------|--------|------|--|--|--|--|
| QBDY1 | SID | Q0   | EID1 | "THRU" | EID2 |  |  |  |  |
| QBDY1 | 109 | 1.-5 | 725  | THRU   | 735  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| Q0         | Heat flux into element. (Real)   |
| EIDi       | CHBDYj element identification numbers. (Integer ≠ 0 or "THRU". For "THRU" option EID2 > EID1.) |

#### Remarks:

1. QBDY1 entries must be selected with the Case Control command LOAD = SID in order to be used in static analysis. The total power into an element is given by the equation:

$$P_{in} = (\text{Effective area}) \cdot Q0$$

2. QBDY1 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. The total power into an element is given by the equation:

$$P_{in}(t) = (\text{Effective area}) \cdot Q0 \cdot F(t - \tau)$$

where the function of time  $F(t - \tau)$  is specified on a TLOADi entry.

3. The sign convention for Q0 is positive for heat input.
4. RC network solver does not support QBDY1 for thermal analysis.

**QBDY2**

## Boundary Heat Flux Load for CHBDYj Elements, Form 2

Defines grid point heat flux into CHBDYj elements.

**Format:**

| 1     | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| QBDY2 | SID | EID | Q01 | Q02 | Q03 | Q04 | Q05 | Q06 |    |
|       | Q07 | Q08 |     |     |     |     |     |     |    |

**Example:**

|       |     |     |      |      |      |      |  |  |  |
|-------|-----|-----|------|------|------|------|--|--|--|
| QBDY2 | 109 | 721 | 1.-5 | 1.-5 | 2.-5 | 2.-5 |  |  |  |
|-------|-----|-----|------|------|------|------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)                                      |
| EID        | Identification number of an CHBDYj element. (Integer > 0)                          |
| Q0i        | Heat flux at the i-th grid point on the referenced CHBDYj element. (Real or blank) |

**Remarks:**

1. QBDY2 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into each point i on an element is given by

$$P_i = AREA_i \cdot Q0i$$

2. QBDY2 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. All connected grid points will have the same time function but may have individual delays. The total power into each point i on an element is given by

$$P_i(t) = AREA_i \cdot Q0i \cdot F(t - \tau_i)$$

where  $F(t - \tau_i)$  is a function of time specified on a TLOADi entry.

3. The sign convention for Q0i is positive for heat flux input to the element.
4. RC network solver does not support QBDY2 for thermal analysis.

**QBDY3****Boundary Heat Flux Load for a Surface**

Defines a uniform heat flux load for a boundary surface.

**Format:**

| 1     | 2    | 3    | 4       | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|------|---------|------|------|------|------|------|----|
| QBDY3 | SID  | Q0   | CNTRLND | EID1 | EID2 | EID3 | EID4 | EID5 |    |
|       | EID6 | etc. |         |      |      |      |      |      |    |

**Example:**

|       |   |      |    |   |      |    |    |   |  |
|-------|---|------|----|---|------|----|----|---|--|
| QBDY3 | 2 | 20.0 | 10 | 1 | THRU | 50 | BY | 2 |  |
|-------|---|------|----|---|------|----|----|---|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| Q0         | Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real) |
| CNTRLND    | Control point for thermal flux load. (Integer $\geq 0$ ; Default = 0)                           |
| EIDi       | CHBDYj element identification numbers. (Integer $\neq 0$ or “THRU” or “BY”)                     |

**Remarks:**

1. QBDY3 entries must be selected in Case Control (LOAD = SID) to be used in steady state. The total power into a surface is given by the equation:
  - if CNTRLND  $\leq 0$  then  $P_{in} = (\text{Effective area}) \cdot Q0$
  - if CNTRLND  $> 0$  then  $P_{in} = (\text{Effective area}) \cdot Q0 \cdot u_{\text{CNTRLND}}$   
where  $u_{\text{CNTRLND}}$  is the temperature of the control point and is used as a load multiplier.
2. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi multiplies the general load, with  $\tau$  specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
3. The CNTRLND multiplier cannot be used with any higher-order elements.
4. When using “THRU” or “BY”, all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.
5. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

## **QHBDY**

### Boundary Heat Flux Load

Defines a uniform heat flux into a set of grid points.

#### **Format:**

| 1     | 2   | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|-----|------|----|----|----|----|----|----|----|
| QHBDY | SID | FLAG | Q0 | AF | G1 | G2 | G3 | G4 |    |
|       | G5  | G6   | G7 | G8 |    |    |    |    |    |

#### **Example:**

|       |   |       |      |  |     |     |     |     |  |
|-------|---|-------|------|--|-----|-----|-----|-----|--|
| QHBDY | 2 | AREA4 | 20.0 |  | 101 | 102 | 104 | 103 |  |
|-------|---|-------|------|--|-----|-----|-----|-----|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification number. (Integer > 0)  |
| FLAG       | Type of face involved (must be one of the following: "POINT", "LINE", "REV", "AREA3", "AREA4", "AREA6", "AREA8") |
| Q0         | Magnitude of thermal flux into face. Q0 is positive for heat into the surface. (Real)                            |
| AF         | Area factor depends on type. (Real > 0.0 or blank)   |
| Gi         | Grid point identification of connected grid points. (Integer > 0 or blank)                                       |

#### **Remarks:**

1. The continuation entry is optional.
2. For use in steady state analysis, the load set is selected in the Case Control Section (LOAD = SID).
3. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
4. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.
5. The flux is applied to each point i by the equation  

$$P_i = AREA_i \cdot Q0$$
6. The number of connected points for the types are 1 (POINT), 2 (LINE, REV), 3 (AREA3), 4 (AREA4), 4-6 (AREA6), 5-8 (AREA8).

7. The area factor AF is used to determine the effective area for the POINT and LINE types. It equals the area and effective width, respectively.. For axisymmetric elements, the magnitude of the area factors should account for an integration over the entire circumference (i.e.,  $2\pi$  radians). Note that the REV option can be used to apply the heat flux along the edges of axisymmetric elements. AF is not used for the other types, which have their area defined implicitly and must be left blank.
8. The type of face (FLAG) defines a surface in the same manner as the CHBDYi Bulk Data entry. For physical descriptions of the geometry involved, see the CHBDYG discussion.

**QSET****Generalized Degree-of-Freedom**

Defines generalized degrees-of-freedom (q-set) to be used for dynamic reduction or component mode synthesis.

**Format:**

| 1    | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|------|-----|----|-----|----|-----|----|-----|----|----|
| QSET | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|      |    |        |   |   |   |   |     |   |  |
|------|----|--------|---|---|---|---|-----|---|--|
| QSET | 15 | 123456 | 1 | 3 | 9 | 2 | 105 | 6 |  |
|------|----|--------|---|---|---|---|-----|---|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

**Remarks:**

- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
- When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
- If Modules are present then this entry may only be specified in the main Bulk Data section.

**QSET1**

## Generalized Degree-of-Freedom (Alternate Form of QSET Entry)

Defines generalized degrees-of-freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.

**Format:**

| 1     | 2   | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|-----|--------|-----|-----|-----|-----|-----|----|
| QSET1 | C   | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|       | ID8 | ID9 | -etc.- |     |     |     |     |     |    |

**Example:**

|       |        |    |   |   |    |     |   |    |  |
|-------|--------|----|---|---|----|-----|---|----|--|
| QSET1 | 123456 | 1  | 7 | 9 | 22 | 105 | 6 | 22 |  |
|       | 52     | 53 |   |   |    |     |   |    |  |

**Alternate Format and Example:**

|       |   |     |        |     |  |  |  |  |  |
|-------|---|-----|--------|-----|--|--|--|--|--|
| QSET1 | C | ID1 | "THRU" | ID2 |  |  |  |  |  |
| QSET1 | 0 | 101 | THRU   | 110 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2.)   |

**Remarks:**

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

**QVECT**

## Thermal Vector Flux Load

Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

**Format:**

| 1     | 2    | 3    | 4      | 5  | 6          | 7          | 8          | 9       | 10 |
|-------|------|------|--------|----|------------|------------|------------|---------|----|
| QVECT | SID  | Q0   | TSOUR  | CE | E1 or TID1 | E2 or TID2 | E3 or TID3 | CNTRLND |    |
|       | EID1 | EID2 | -etc.- |    |            |            |            |         |    |

**Example:**

|       |    |      |        |    |     |     |     |     |  |
|-------|----|------|--------|----|-----|-----|-----|-----|--|
| QVECT | 10 | 20.0 | 1000.0 |    | 1.0 | 1.0 | 1.0 | 101 |  |
|       | 20 | 21   | 22     | 23 |     |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| Q0         | Magnitude of thermal flux vector into face. (Real or blank)   |
| TSOUR      | Temperature of the radiant source. (Real or blank)  |
| CE         | Coordinate system identification number for thermal vector flux. See Remark 9.<br>(Integer $\geq -1$ or blank)  |
| Ei         | Vector components (direction cosines in coordinate system CE) of the thermal vector flux. (Real; Default = 0.0)   |
| TIDI       | TABLEDi entry identification numbers defining the components as a function of time. (Integer > 0)   |
| CNTRLND    | Control point. (Integer $\geq 0$ ; Default = 0)   |
| EIDI       | Element identification number of a CHBDYE, CHBDYG, or CHBDYP entry.<br>(Integer > 0) Key word "THRU" can be used to assist the listing with ascending order of EIDI |

**Remarks:**

1. The continuation entry is required.
2. If the coordinate system CE is not rectangular, then the thermal vector flux is in different directions for different CHBDYi elements. The direction of the thermal vector flux over an element is aligned to be in the direction of the flux vector at the geometric center of the element. The geometric center is measured using the grid points and includes any DISLIN specification on the VIEW entry for TYPE=LINE CHBDYi elements. The flux is presumed to be uniform over the face of each element; i.e., the source is relatively distant.

3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID). The total power into an element is given by:

- If CNTRLND = 0 then,  $P_{in} = -\alpha A(\hat{e} \cdot \hat{n}) \cdot Q0$ .
- If CNTRLND > 0 then,  $P_{in} = -\alpha A(\hat{e} \cdot \hat{n}) \cdot Q0 \cdot u_{CNTRLND}$ .

where

$\alpha$  = face absorptivity (supplied from a RADM statement).

$A$  = face area as determined from a CHBDYi surface element.

$\hat{e}$  = vector of direction cosines E1, E2, E3.

$\hat{n}$  = face normal vector. See CHBDYi entries.

$\hat{e} \cdot \hat{n}$  = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

$u_{cntrlnd}$  = temperature value of the control point used as a load multiplier.

4. If the absorptivity is constant, its value is supplied by the ABSORP field on the RADM entry. If the absorptivity is not a constant, the thermal flux is assumed to have a wavelength distribution of a black body at the temperature TSOUR.

- For a temperature-dependent absorptivity, the element temperature is used to determine  $\alpha$ .
- For a wavelength-dependent absorptivity, the integration of the flux times  $\alpha$  is computed for each wavelength band. The sum of the integrated thermal fluxes over all the wavelength bands is Q0. The wave bands are specified with the RADBND entry.
- The user has the responsibility of enforcing Kirchhoff's laws.

5. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  provides any required time delay.  $F(t - \tau)$  is a function of time specified on the TLOADi entry. The value of  $F$  is calculated for each loaded grid point. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

The total power into an element is given by:

- If CNTRLND = 0 then,  $P_{in} = -\alpha A(\hat{e}(t) \cdot \hat{n}) \cdot Q0 \cdot F(t - \tau)$ .
- If CNTRLND > 0 then,

$$P_{in} = -\alpha A(\hat{e}(t) \cdot \hat{n}) \cdot F(t - \tau) \cdot Q0 \cdot u_{CNTRLND}.$$

6. If the referenced face is of TYPE = ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.

7. If the referenced face is of TYPE = REV, the thermal flux vector must be parallel to the axis of symmetry if an axisymmetric boundary condition is to be maintained.

8. When applied to a surface element associated with a radiation enclosure cavity, any incident energy that is not absorbed ( $\alpha < 1.0$ ) is lost from the system and is not accounted for in a reflective sense ( $\alpha + \rho = 1.0$ ).

9. If a heat flux normal to the surface is desired, set CE to -1. This allows a nondirectional temperature dependent heat load on the CHBDYi. the RADMT scale factor times Q0 equals to the total power. Remember that the absorptivity must fall between 0.0 and 1.0, (see the RADMT Bulk Data entry).
10. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

**QVOL****Volume Heat Addition**

Defines a rate of volumetric heat addition in a conduction element.

**Format:**

| 1    | 2    | 3    | 4       | 5    | 6    | 7    | 8    | 9    | 10 |
|------|------|------|---------|------|------|------|------|------|----|
| QVOL | SID  | QVOL | CNTRLND | EID1 | EID2 | EID3 | EID4 | EID5 |    |
|      | EID6 | etc. |         |      |      |      |      |      |    |

**Example:**

|      |   |      |     |    |    |    |   |  |  |
|------|---|------|-----|----|----|----|---|--|--|
| QVOL | 5 | 10.0 | 101 | 10 | 12 | 11 | 9 |  |  |
|------|---|------|-----|----|----|----|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Load set identification. (Integer > 0)   |
| QVOL       | Power input per unit volume produced by a heat conduction element. (Real)            |
| CNTRLND    | Control point used for controlling heat generation. (Integer $\geq 0$ ; Default = 0) |
| EIDi       | A list of heat conduction elements. (Integer > 0 or “THRU” or “BY”)                  |

**Remarks:**

1. EIDi has material properties (MAT4) that include HGEN, the element material property for heat generation, which may be temperature dependent. This association is made through the element EID. If HGEN is temperature dependent, it is based on the average element temperature.
2. QVOL provides either the constant volumetric heat generation rate or the load multiplier. QVOL is positive for heat generation. For steady-state analysis, the total power into an element is
  - If CNTRLND = 0, then  $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL}$  .
  - If CNTRLND > 0, then  $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL} \cdot u_{\text{CNTRLND}}$  .
where  $u_{\text{CNTRLND}}$  is the temperature multiplier.
3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID).
4. In transient analysis SID is referenced by a TLOADi Bulk Data entry. A function of time  $F[t - \tau]$  defined on the TLOADi entry multiplies the general load where  $\tau$  specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
5. For “THRU” or “BY”, all intermediate referenced heat conduction elements must exist.
6. The CNTRLND multiplier cannot be used with any higher-order elements.
7. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

8. QVOL is not supported for CQUADX and CTRIAX axisymmetric elements. It is supported for the CTRIA6 element.

**RADBC**

## Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

**Format:**

| 1     | 2      | 3    | 4       | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|--------|------|---------|------|------|------|------|------|----|
| RADBC | NODAMB | FAMB | CNTRLND | EID1 | EID2 | EID3 | EID4 | EID5 |    |
|       | EID6   | EID7 | -etc.-  |      |      |      |      |      |    |

**Example:**

|       |   |     |     |    |  |  |  |  |  |
|-------|---|-----|-----|----|--|--|--|--|--|
| RADBC | 5 | 1.0 | 101 | 10 |  |  |  |  |  |
|-------|---|-----|-----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| NODAMB     | Ambient point for radiation exchange. (Integer > 0)  |
| FAMB       | Radiation view factor between the face and the ambient point. (Real $\geq 0.0$ )   |
| CNTRLND    | Control point for radiation boundary condition. (Integer $\geq 0$ ; Default = 0)   |
| EIDi       | CHBDYi element identification number. (Integer > 0) Key words "THRU" and "BY" can be used to assist the listing with ascending order of EIDi |

**Remarks:**

1. The basic exchange relationship is:
  - if CNTRLND = 0, then  $q = \sigma \cdot FAMB \cdot \epsilon_e \cdot (T_e^4 - T_{amb}^4)$
  - if CNTRLND > 0, then  

$$q = \sigma \cdot FAMB \cdot u_{CNTRLND} \cdot \epsilon_e \cdot (T_e^4 - T_{amb}^4)$$
2. NODAMB is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space. No surface element that is a member of a radiation enclosure cavity may also have a radiation boundary condition applied to it.
3. Two PARAM entries are required when stipulating radiation heat transfer:
  - ABS defines the absolute temperature scale; this value is added internally to any specified temperature given in the problem. Upon solution completion, this value is subtracted internally from the solution vector.
  - SIGMA ( $\sigma$ ) is the Stefan-Boltzmann constant.
4. RADBC allows for surface radiation to space in the absence of any cavity behavior. The emissivity is supplied from a RADM entry.
5. When using "THRU" or "BY", all intermediate referenced CHBDYi surface elements must exist.

**RADBND****Radiation Wavelength Band Definition**

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

**Format:**

| 1      | 2       | 3       | 4       | 5       | 6       | 7       | 8       | 9       | 10 |
|--------|---------|---------|---------|---------|---------|---------|---------|---------|----|
| RADBND | NUMBER  | PLANCK2 | LAMBDA1 | LAMBDA2 | LAMBDA3 | LAMBDA4 | LAMBDA5 | LAMBDA6 |    |
|        | LAMBDA7 | etc.    |         |         |         |         |         |         |    |

**Example:**

|        |   |         |     |     |     |     |      |  |  |
|--------|---|---------|-----|-----|-----|-----|------|--|--|
| RADBND | 6 | 14388.0 | 1.0 | 2.0 | 4.0 | 8.0 | 12.0 |  |  |
|--------|---|---------|-----|-----|-----|-----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| NUMBER     | Number of radiation wave bands. See Remarks. (Integer > 1)               |
| PLANCK2    | Planck's second radiation constant. See Remarks. (Real > 0.0)            |
| LAMBDAi    | Highest wavelength of the i-th wave band. See Remarks. (Real $\geq$ 0.0) |

**Remarks:**

- Only one RADBND entry may be specified in the Bulk Data Section and must always be used in conjunction with the RADM entry.
- PLANCK2 has the units of wavelength times temperature. The same units of length must be used for LAMBDAi as for PLANCK2. The units of temperature must be the same as those used for the radiating surfaces. For example:  $25898.\mu\text{m } ^\circ\text{R}$  or  $14388.\mu\text{m } ^\circ\text{K}$ .
- The first wavelength band extends from 0 to LAMBDA1 and the last band extends from LAMBDA $n$  to infinity, where  $n = \text{NUMBER} - 1$ .
- Discontinuous segments in the emissivity versus wavelength piecewise linear curve must be treated as a wavelength band of zero width.
- LAMBDAi must be specified in ascending order, and all LAMBDAi fields where i is greater than or equal to NUMBER must be blank.
- If Modules are present then this entry may only be specified in the main Bulk Data section.

**RADC**

## Thermal Radiative Coating Properties - SOL400 - RC Network solver

Defines the radiative properties of advanced materials such as coatings and multilayer insulation, commonly used in the aerospace market.

**Format: (COAT)**

| 1    | 2      | 3         | 4            | 5          | 6           | 7             | 8             | 9 | 10 |
|------|--------|-----------|--------------|------------|-------------|---------------|---------------|---|----|
| RADC | RADMID | Emis      | Absorptivity | IR Spec    | UV Spec     |               |               |   |    |
|      | "COAT" | IR TraspA | IR Transluc  | UV TranspA | UV Transluc | IR Refrac Ind | UV Refrac Ind |   |    |

**Example:**

|      |      |    |    |    |    |    |    |  |  |
|------|------|----|----|----|----|----|----|--|--|
| RADC | 101  | 1. | 1. | 0. |    |    |    |  |  |
|      | COAT | 0. | 0. | 0. | 0. | 1. | 1. |  |  |

**Format: (MLI)**

| 1    | 2      | 3     | 4            | 5       | 6       | 7 | 8 | 9 | 10 |
|------|--------|-------|--------------|---------|---------|---|---|---|----|
| RADC | RADMID | Emis  | Absorptivity | IR Spec | UV Spec |   |   |   |    |
|      | "MLI"  | Estar |              |         |         |   |   |   |    |

**Example:**

|      |     |      |    |    |  |  |  |  |  |
|------|-----|------|----|----|--|--|--|--|--|
| RADC | 102 | 1.   | 1. | 0. |  |  |  |  |  |
|      | MLI | 0.02 |    |    |  |  |  |  |  |

| Descriptor    | Meaning  |
|---------------|--|
| RADMID        | Radiation material identification number (Integer > 0; Required)         |
| COAT          | Identification that a coating type material has been started (Character) |
| Emis          | Emissivity (Real $\geq$ 0.0; Default 1.0)                                |
| Absorptivity  | Absorptivity (Real $\geq$ 0.0; Default 1.0)                              |
| IR Spec       | IR Specularity (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                 |
| UV Spec       | UV Specularity (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                 |
| IR TranspA    | IR Transparency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                |
| IR Transluc   | IR Translucency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                |
| UV TranspA    | UV Transparency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                |
| UV Transluc   | UV Translucency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)                |
| IR Refrac Ind | IR Refraction Index (Real $\geq$ 1.0; Default 1.0)                       |
| UV Refrac Ind | UV Refraction Index (Real $\geq$ 1.0; Default 1.0)                       |
| MLI           | Identification that an MLI type material has been started (Character)    |
| Estar         | MLI E-star parameter (Real; Default 0.0)                                 |

**Remarks:**

1. This entry is for RC Network solver only.
2. Estar is defined as the effective emissivity from the wall to MLI outer surface. The general value is around 0.01 to 0.03, the typical value is 0.02.
3. Emis is usually for the IR waveband, and Absorptivity is for the UV waveband.

**RADCAV****Radiation Cavity Identification**

Identifies the characteristics of each radiant enclosure.

**Format:**

| 1      | 2       | 3      | 4      | 5     | 6      | 7     | 8    | 9     | 10 |
|--------|---------|--------|--------|-------|--------|-------|------|-------|----|
| RADCAV | ICAVITY | ELEAMB | SHADOW | SCALE | PRTPCH | NFECI | RMAX | NCOMP |    |
|        | SET11   | SET12  | SET21  | SET22 | SET31  | SET32 | etc. |       |    |

**Example:**

|        |   |   |   |   |   |   |     |  |  |
|--------|---|---|---|---|---|---|-----|--|--|
| RADCAV | 1 | 1 |   |   |   |   | .99 |  |  |
|        | 3 | 5 | 4 | 5 | 7 | 5 |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| ICAVITY    | Unique cavity identification number associated with enclosure radiation. (Integer > 0)   |
| ELEAMB     | CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.) |
| SHADOW     | Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character = "YES" or "NO"; Default = "YES")           |
| SCALE      | View factor that the enclosure sum will be set to if a view factor is greater than 1.0. (0.0 ≤ Real ≤ 1.0; Default = 0.0)  |
| PRTPCH     | Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer = 0, 1, 2, 3, 4, or 5; Default = blank):                  |

| Hemi-Cube & VIEW3D |                       |                       |
|--------------------|-----------------------|-----------------------|
| Value              | Printout in .f06 file | Printout in .pch file |
| Blank              | No                    | Yes                   |
| 0                  | Full Print            | Yes                   |
| 1                  | No                    | Yes                   |
| 2                  | Full Print            | No                    |
| 3                  | No                    | No                    |
| 4                  | Summary Print         | Yes                   |
| 5                  | Summary Print         | No                    |

|       |  |
|-------|--|
| NFECI | Controls whether finite difference or contour integration methods are to be used in the calculation of view factors in the absence of a VIEW3D Bulk Data entry. (Character = “FD” or “CONT”; See Remark 4. for default.) |
| RMAX  | Subelement area factor. See Remark 5. (Real $\geq 0.0$ ; Default = 0.1)  |
| NCOMP | Total number of computational element for one-half ring. See Remark 8. (Default = 32)  |
| SETij | Set identification pairs for the calculation of global view factors. Up to 30 pairs may be specified (i = 1 to 2 and j = 1 to 30). (Integer $> 0$ )  |

### Remarks:

1. For the surfaces of an incomplete enclosure (view factors add up to less than 1.0), a complete enclosure may be achieved ( $\text{SUM} = 1.0$ ) by specifying an ambient element, ELEAMB. When multiple cavities are defined, each cavity must have a unique ambient element if ambient elements are desired. No elements can be shared between cavities.
2. Third-body shadowing is ignored in the cavity if SHADOW = “NO”. In particular, if it is known a priori that there is no third-body shadowing, SHADOW = NO overrides KSHD and KBSD fields on the VIEW Bulk Data entry as well as reduces the calculation time immensely.
3. The view factors for a complete enclosure may add up to slightly more than 1.0 due to calculation inaccuracies. SCALE can be used to adjust all the view factors proportionately to acquire a summation equal to the value specified for SCALE. If SCALE is left blank or set to 0.0, no scaling is performed.
4. If the VIEW3D Bulk Data entry is not specified, the view factors are calculated using finite difference and contour integration methods. If NFECI = “FD”, then all view factors are calculated using the finite difference technique. NFECI = “CONT” invokes contour integration for all view factor calculations. If NFECI is blank, the program selects a method to use between any two particular elements based on RMAX.
5. The comparison value for RMAX is equal to  $A_s/d_{rs}^2$  where  $A_s$  is the area of a subelement and  $d_{rs}$  is the distance between two subelements r and s for which view factors are being computed. When NFECI is blank, the program selects the contour integral method only if  $A_s/d_{rs}^2 > \text{RMAX}$ .
6. When a number of elements are grouped together and considered as a conglomerate surface, view factors can be calculated between these groups. These are referred to as global view factors. The SET1 Bulk Data entry is used to define the conglomerate. When using this feature, negative EIDs are not allowed.
7. If a RADLST and RADMTX entry exists for this cavity ID, new view factors are not computed and the existing RADLST and RADMTX are used in the thermal analysis.
8. The VIEW3D Bulk Data entry must be specified for the calculation of axisymmetric view factors. The process relies on the internal construction of a semi-circle of computational elements. NCOMP specifies the number of such elements desired.
9. For SOL 600, fields 4-8 of the primary entry and all continuation lines are ignored.

10. For SOL 600 field 9 of the primary entry (NCOMP) is used to indicate if the cavity is open or closed according to the following designation:
  - 0 Cavity is closed
  - 1 Cavity is closed and view factors are scaled such that they sum to exactly 1.0
  - 2 Cavity is open
11. For RC network solver in thermal analysis, RADCAV is only used to specify radiation space node, the ELEAMB must be a POINT type CHBDYP element. The SHADOW, SCALE, PRTPCH, NFECI, RMAX, NCOMP, SET11, SET12, SET21, SET22, SET31 and SET32 are ignored.

**RADCOL**

Radiation Collective Entity - SOL400 - RC Network solver

Specifies a collection of boundary elements to be used as a single face in the radiation calculation. This will decrease computation time at the small cost of accuracy. Computational savings and accuracy are dependent on the coarseness of the collection versus the constituents. View factors of the collection are redistributed across the elements for calculation of the radiative energy transfer.

**Format:**

| 1      | 2        | 3      | 4      | 5       | 6       | 7      | 8 | 9 | 10 |
|--------|----------|--------|--------|---------|---------|--------|---|---|----|
| RADCOL | RADCOLID | IVIEWF | IVIEWB | RADMIDF | RADMIDB | SET3ID |   |   |    |

**Example:**

|        |     |   |   |   |   |   |  |  |  |
|--------|-----|---|---|---|---|---|--|--|--|
| RADCOL | 101 | 5 | 6 | 2 | 3 | 7 |  |  |  |
|--------|-----|---|---|---|---|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| RADCOLID   | Radiation Collection identification number. (Integer > 0; Required)                   |
| IVIEWF     | A VIEW entry identification number for the front face. (Integer $\geq 0$ ; Default 0) |
| IVIEWB     | A VIEW entry identification number for the back face. (Integer $\geq 0$ ; Default 0)  |
| RADMIDF    | RADM identification number for the front face. (Integer $\geq 0$ ; Default 0)         |
| RADMIDB    | RADM identification number for the back face. (Integer $\geq 0$ ; Default 0)          |
| SET3 idn   | ID of the element collection to be considered a super element. (Integer > 0)          |

**Remarks:**

1. This entry is for RC Network solver only.
2. IVIEWF/IVIEWB will default to 0 if left blank. This would indicate that the corresponding front and/or back do not participate in the radiation.
3. If an IVIEWF is specified, there must also be a RADMIDF for surface material properties. If a IVIEWB is specified, there must also be a RADMIDB for surface material properties.

## RADCT      Thermal Radiative Functionally-Dependent Properties - SOL400 - RC Network solver

Specifies table references for RADCT properties that are functionally dependent.

### Format:

| 1     | 2      | 3         | 4        | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|--------|-----------|----------|---|---|---|---|---|----|
| RADCT | RADMID | Emis f(T) | Abs f(T) |   |   |   |   |   |    |

### Example:

|       |    |     |     |  |  |  |  |  |  |
|-------|----|-----|-----|--|--|--|--|--|--|
| RADCT | 11 | 101 | 102 |  |  |  |  |  |  |
|-------|----|-----|-----|--|--|--|--|--|--|

| Descriptor        | Meaning  |
|-------------------|--|
| RADMID            | Radiation material identification number (Integer > 0; Required) |
| Emis f(T)         | Emissivity TABLEMj table ID (Integer $\geq 0$ ; Default.0)       |
| Absorptivity f(T) | Absorptivity TABLEDj table ID (Integer $\geq 0$ ; Default 0)     |

### Remarks:

1. This entry is for RC Network solver only.
2. RADCT always comes with the RADCT entry. They have the same RADMID so that they can be referenced together.
3. Currently, RC Network solver does not support this entry for enclosure radiation.

**RADLST****Listing of Enclosure Radiation Faces**

Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.

**Format:**

| 1      | 2       | 3      | 4    | 5    | 6    | 7    | 8    | 9    | 10 |
|--------|---------|--------|------|------|------|------|------|------|----|
| RADLST | ICAVITY | MTXTYP | EID1 | EID2 | EID3 | EID4 | EID5 | EID6 |    |
|        | EID7    | -etc.- |      |      |      |      |      |      |    |

**Example:**

|        |   |   |   |   |   |   |   |   |  |
|--------|---|---|---|---|---|---|---|---|--|
| RADLST | 2 | 1 | 2 | 3 | 4 | 5 | 6 | 7 |  |
|--------|---|---|---|---|---|---|---|---|--|

| Descriptor | Meaning  |
|------------|--|
| ICAVITY    | Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation faces. (Integer > 0)  |
| MTXTYP     | Type of radiation exchange matrix used for this cavity. (Integer $\leq 4$ and $\neq 0$ ; Default = 1 for an enclosure without an ambient element. Default = 4 for an enclosure with an ambient element as specified on the RADCAV entry.)  |
|            | 1 Symmetric view factor matrix [F] and nonconservative radiation matrix [R].<br>2 Symmetric exchange factor matrix [ $\mathfrak{J}$ ] and conservative radiation matrix [R].<br>3 Unsymmetric exchange factor matrix [ $\mathfrak{J}$ ] and conservative radiation matrix [R].<br>4 Symmetric view factor matrix [F] and conservative radiation matrix [R].<br>-n The first n CHBDYi elements may lose energy to space but the remainder may not. Symmetric exchange factor matrix [F] and nonconservative radiation matrix [R]. |
| EIDI       | Identification numbers for the CHBDYi elements in this radiation cavity. (Integer $\neq 0$ or "THRU")  |

**Remarks:**

1. A radiation EIDI list isolates those CHBDYi surface element faces that are to communicate in a radiation enclosure. View-factor calculation and RADMTX formation for an enclosure is performed only for (or among) those faces identified within the same RADCAV.
2. A radiation exchange matrix (RADMTX) can only reference one radiative face list (RADLST). The companion RADCAV, RADLST, and RADMTX must share a unique ICAVITY.
3. For each EIDI, the appropriate CHBDYi element is located, and the proper RADM entry ID field found.
4. If the radiation exchange matrix or any radiation boundary conditions are available from an external source, the RADMTX must be user generated.

5. Multiple RADLST entries may be specified.
6. If any RADLST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADLST entry changes or additions are not recommended on restart.
7. RC network solver does not support RADLST for thermal analysis.

**RADM****Radiation Boundary Material Property**

Defines the radiation properties of a boundary element for heat transfer analysis.

**Format:**

| 1    | 2      | 3      | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|------|--------|--------|-------|-------|-------|-------|-------|-------|----|
| RADM | RADMID | ABSORP | EMIS1 | EMIS2 | EMIS3 | EMIS4 | EMIS5 | EMIS6 |    |
|      | EMIS7  | -etc.- |       |       |       |       |       |       |    |

**Example:**

|      |    |  |     |     |     |     |     |     |  |
|------|----|--|-----|-----|-----|-----|-----|-----|--|
| RADM | 11 |  | .45 | .33 | .29 | .20 | .17 | .13 |  |
|------|----|--|-----|-----|-----|-----|-----|-----|--|

| Descriptor | Meaning  |
|------------|--|
| RADMID     | Material identification number. (Integer > 0)  |
| ABSORP     | Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2. ( $0.0 \leq \text{Real} \leq 1.0$ )                             |
| EMISi      | Surface emissivity at wavelength LAMBDAi or the temperature function curve multiplier if EMISi is variable (See the RADBND entry.) ( $0.0 \leq \text{Real} \leq 1.0$ ) |

**Remarks:**

1. The RADM entry is directly referenced only by one of the CHBDYE, CHBDYG, or CHBDYP type surface element entries.
2. For radiation enclosure problems, ABSORP is set equal to emissivity. For QVECT loads, absorptivity is specified by ABSORP.
3. If there is more than one EMISi, then:
  - There must be a RADBND entry.
  - The number of EMISi may not exceed the number of LAMBDAi on the RADBND entry.
  - The emissivity values are given for a wavelength specified by the corresponding LAMBDAi on the RADBND entry. Within each discrete wavelength band, the emissivity is assumed to be constant.
  - At any specific wavelength and surface temperature, the absorptivity is exactly equal to the emissivity.
4. To perform any radiation heat transfer exchange, the user must furnish PARAM entries for:
  - TABS to define the absolute temperature scale.
  - SIGMA ( $\sigma$ ) to define the Stefan-Boltzmann constant in appropriate units.
5. RC network solver only supports RADMID, ABSORP and EMIS1 for thermal analysis, other fields are ignored.

**RADMT****Radiation Boundary Material Property Temperature Dependence**

Specifies table references for temperature dependent RADM entry radiation boundary properties.

**Format:**

| 1     | 2      | 3                    | 4                    | 5                    | 6                    | 7                    | 8                    | 9                    | 10 |
|-------|--------|----------------------|----------------------|----------------------|----------------------|----------------------|----------------------|----------------------|----|
| RADMT | RADMID | T(A)                 | T( $\varepsilon_1$ ) | T( $\varepsilon_2$ ) | T( $\varepsilon_3$ ) | T( $\varepsilon_4$ ) | T( $\varepsilon_5$ ) | T( $\varepsilon_6$ ) |    |
|       |        | T( $\varepsilon_7$ ) | -etc.-               |                      |                      |                      |                      |                      |    |

**Example:**

|       |    |  |   |   |   |   |   |   |  |
|-------|----|--|---|---|---|---|---|---|--|
| RADMT | 11 |  | 1 | 2 | 3 | 4 | 5 | 6 |  |
|-------|----|--|---|---|---|---|---|---|--|

| Descriptor           | Meaning  |
|----------------------|--|
| RADMID               | Material identification number. (Integer > 0)                            |
| T(A)                 | TABLEMj identifier for surface absorptivity. (Integer $\geq 0$ or blank) |
| T( $\varepsilon_i$ ) | TABLEMj identifiers for surface emissivity. (Integer $\geq 0$ or blank)  |

**Remarks:**

1. The basic quantities on the RADM entry of the same RADMID are always multiplied by the corresponding tabular function.
2. Tables T(A) and T( $\varepsilon_i$ ) have an upper bound that is less than or equal to one and a lower bound that is greater than or equal to zero.
3. The TABLEMj enforces the element temperature as the independent variable. Blank or zero fields means there is no temperature dependence of the referenced property on the RADM entry.
4. RC network solver only supports RADMID, T(A) and T( $\varepsilon_1$ ) for thermal analysis, other fields are ignored.

**RADMTX****Radiation Exchange Matrix**

Provides the  $F_{ji} = A_j f_{ji}$  exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.

**Format:**

| 1      | 2       | 3      | 4    | 5      | 6      | 7      | 8      | 9      | 10 |
|--------|---------|--------|------|--------|--------|--------|--------|--------|----|
| RADMTX | ICAVITY | INDEX  | Fi,j | Fi+1,j | Fi+2,j | Fi+3,j | Fi+4,j | Fi+5,j |    |
|        | Fi+6,j  | -etc.- |      |        |        |        |        |        |    |

**Example:**

|        |   |   |     |     |     |     |     |     |  |
|--------|---|---|-----|-----|-----|-----|-----|-----|--|
| RADMTX | 2 | 1 | 0.0 | 0.1 | 0.2 | 0.2 | 0.3 | 0.2 |  |
|--------|---|---|-----|-----|-----|-----|-----|-----|--|

| Descriptor       | Meaning  |
|------------------|--|
| ICAVITY          | Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation surface elements. (Integer > 0)   |
| INDEX            | Column number in the matrix. (Integer > 0)   |
| F <sub>k,j</sub> | If symmetric, the matrix values start on the diagonal (i = j) and continue down the column (k = i + 1, i + 2, etc.). If unsymmetric, the values start in row (i = 1). i refers to EIDi on the RADLST entry. (Real $\geq 0$ ) |

**Remarks:**

1. If the matrix is symmetric, only the lower triangle is input, and i = j = INDEX. If the matrix is unsymmetric, i = 1, and j = INDEX.
2. Only one ICAVITY may be referenced for those faces that are to be included in a unique radiation matrix.
3. Coefficients are listed by column with the number of columns equal to the number of entries in the RADLST.
4. All faces involved in any radiation enclosure must be defined with an CHBDYi element.
5. If any RADMTX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADMTX entry changes or additions are not recommended on restart.
6. Set NASTRAN SYSTEM (87) = 3 is a new option in MSC Nastran 2005 that prevents radiation energy from being lost to space.
7. RC network solver does not support RADMTX for thermal analysis.

**RADSET**

Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

**Format:**

| 1      | 2        | 3        | 4        | 5        | 6        | 7        | 8        | 9        | 10 |
|--------|----------|----------|----------|----------|----------|----------|----------|----------|----|
| RADSET | ICAVITY1 | ICAVITY2 | ICAVITY3 | ICAVITY4 | ICAVITY5 | ICAVITY6 | ICAVITY7 | ICAVITY8 |    |
|        | ICAVITY9 | -etc.-   |          |          |          |          |          |          |    |

**Example:**

|        |   |   |   |   |  |  |  |  |  |
|--------|---|---|---|---|--|--|--|--|--|
| RADSET | 1 | 2 | 3 | 4 |  |  |  |  |  |
|--------|---|---|---|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ICAVITYi   | Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0) |

**Remark:**

- For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

**RANDPS****Power Spectral Density Specification**

Defines load set power spectral density factors for use in random analysis having the frequency dependent form

$$S_{jk}(F) = (X + iY)G(F)$$

**Format:**

| 1      | 2   | 3 | 4 | 5 | 6 | 7   | 8 | 9 | 10 |
|--------|-----|---|---|---|---|-----|---|---|----|
| RANDPS | SID | J | K | X | Y | TID |   |   |    |

**Example:**

|        |   |   |   |     |     |   |  |  |  |
|--------|---|---|---|-----|-----|---|--|--|--|
| RANDPS | 5 | 3 | 7 | 2.0 | 2.5 | 4 |  |  |  |
|--------|---|---|---|-----|-----|---|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Random analysis set identification number. See Remarks 1 and 9. (Integer > 0)                                    |
| J          | Subcase identification number of the excited load set. See Remarks 6 through 9. (Integer > 0)                    |
| K          | Subcase identification number of the applied load set. See Remarks 6 through 9. (Integer $\geq 0$ ; K $\geq J$ ) |
| X, Y       | Components of the complex number. See Remarks 2 and 7. (Real)  |
| TID        | Identification number of a TABRNDi entry that defines G(F). See Remarks 3 and 7. (Integer > 0 or Blank)          |

**Remarks:**

- Set identification numbers must be selected with the Case Control command (RANDOM = SID) or through use of FTGLOAD when doing random vibration fatigue analysis.
- For auto spectral density, J = K, X must be greater than zero and Y must be equal to zero.
- For TID = Blank, G(F) = 1.0.
- RANDPS may only reference subcases included within a single loop (a change in direct matrix input is not allowed).
- See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
- In the case of cyclic symmetry Solution Sequence 118, J and K must refer to the coded subcase IDs.
- In superelement analysis, J and K must refer to the superelement subcases. For example, if superelement 10 has SUBCASEs 1 and 2 and superelement 20 has SUBCASEs 3 and 4, then a separate RANDPS entry is required for each superelement, even though X, Y, and TID may be identical.
- For uncoupled PSDF (no J < K entries) only one J = K entry is allowed for unique value of J. For coupled PSDF (some J < K entries) any number of entries are allowed.

9. J and K must reference valid subcase IDs. In superelement analysis, J and K must reference valid subcase IDs that pertain to the applicable superelement. If this requirement is not met, the program issues an appropriate user warning message and ignores the associated RANDOM=SID command.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RANDT1****Autocorrelation Function Time Lag**

Defines time lag constants for use in random analysis autocorrelation function calculation.

**Format:**

| 1      | 2   | 3 | 4  | 5    | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|----|------|---|---|---|---|----|
| RANDT1 | SID | N | T0 | TMAX |   |   |   |   |    |

**Example:**

|        |   |    |     |     |  |  |  |  |  |
|--------|---|----|-----|-----|--|--|--|--|--|
| RANDT1 | 5 | 10 | 3.2 | 9.6 |  |  |  |  |  |
|--------|---|----|-----|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Random analysis set identification number. (Integer > 0) |
| N          | Number of time lag intervals. (Integer > 0)              |
| T0         | Starting time lag. (Real $\geq 0.0$ )                    |
| TMAX       | Maximum time lag. (Real $> T0$ )                         |

**Remarks:**

1. Time lag sets must be selected with the Case Control command RANDOM = SID.
  2. At least one RANDPS entry must be present with the same set identification number.
  3. The time lags defined on this entry are given by
- $$T_i = T0 + \frac{TMAX - T0}{N}(i - 1), i = 1, N + 2$$
4. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
  5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RBAR****Rigid Bar**

Defines a rigid bar with six degrees-of-freedom at each end.

**Format:**

| 1    | 2   | 3  | 4  | 5   | 6   | 7   | 8   | 9     | 10 |
|------|-----|----|----|-----|-----|-----|-----|-------|----|
| RBAR | EID | GA | GB | CNA | CNB | CMA | CMB | ALPHA |    |

**Example:**

|      |   |   |   |        |  |  |  |       |  |
|------|---|---|---|--------|--|--|--|-------|--|
| RBAR | 5 | 1 | 2 | 123456 |  |  |  | 6.5-6 |  |
|------|---|---|---|--------|--|--|--|-------|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. ( $0 < \text{Integer} < 100,000,000$ )   |
| GA, GB     | Grid point identification number of connection points. ( $\text{Integer} > 0$ )   |
| CNA, CNB   | Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark 3. (Integers 1 through 6 with no embedded blanks, or zero or blank.)               |
| CMA, CMB   | Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 4. and 5. (Integers 1 through 6 with no embedded blanks, or zero or blank.) |
| ALPHA      | Thermal expansion coefficient. See Remark 11. (Real or blank)   |

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create the Lagrange multiplier degrees-of-freedom internally in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- For the linear method, the total number of components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA = 123456 or CNB = 123456 is allowed. If both CNA and CNB are blank, then CNA = 123456. For this method, RBAR1 gives the simpler input format.
- If both CMA and CMB are zero or blank, all of the degrees-of-freedom not in CNA and CNB will be made dependent. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected in the RIGID Case Control command. However, the rules regarding the m-set described below apply to both methods.

5. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RBAR, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.  
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBAR.
12. When there is large rotation, CMA and CMB must have all “456” if any rotational degrees of freedom is used.

**RBAR1**

Rigid Bar

Alternative format for RBAR.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6     | 7 | 8 | 9 | 10 |
|-------|-----|----|----|----|-------|---|---|---|----|
| RBAR1 | EID | GA | GB | CB | ALPHA |   |   |   |    |

**Example:**

|       |   |   |   |     |       |  |  |  |  |
|-------|---|---|---|-----|-------|--|--|--|--|
| RBAR1 | 5 | 1 | 2 | 123 | 6.5-6 |  |  |  |  |
|-------|---|---|---|-----|-------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (0 < Integer < 100,000,000)  |
| GA, GB     | Grid point identification numbers. (Integer > 0)  |
| CB         | Component numbers in the global coordinate system at GB, which are constrained to move as the rigid bar. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.) |
| ALPHA      | Thermal expansion coefficient. See Remark 8. (Real or blank)  |

**Remarks:**

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
3. RBAR1 is a preferred input format to define the Lagrange method for a rigid bar.
4. When CB = "123456" or blank, the grid point GB is constrained to move with GA as a rigid bar. For default CB = "123456". Any number of degrees-of-freedom at grid point GB can be released not to move with the rigid body.
5. For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA and GB will be computed even if these two points have different global coordinate systems.
6. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBAR1.

7. Element identification numbers should be unique with respect to all other element identification numbers.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

**RBAX3D**

## Axisymmetric Harmonic Element Rigid Connection

Defines a rigid connection between a point on an axisymmetric harmonic element (CQUADX or CTRIAX) and a point on the axis of symmetry of that element.

**Format:**

| 1      | 2   | 3   | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-----|-----|---|---|---|---|---|----|
| RBAX3D | EID | G3D | GAX |   |   |   |   |   |    |

**Example:**

|        |     |    |    |  |  |  |  |  |  |
|--------|-----|----|----|--|--|--|--|--|--|
| RBAX3D | 100 | 10 | 20 |  |  |  |  |  |  |
|--------|-----|----|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000). See Remark 1..   |
| G3D        | Identification number of a grid point on the axis of symmetry (Integer > 100,000,000). See Remarks 2. and 4. through 7.  |
| GAX        | Identification number of a grid point referenced on an axisymmetric harmonic element (CQUADX or CTRIAX). (0 < Integer < 100,000,000). See Remarks 3. through 7.. |

**Remarks:**

1. The EID should be unique with respect to all other elements.
2. The G3D point may not be referenced on an axisymmetric harmonic element. If it is, the program terminates the execution with an appropriate user fatal message.
3. The GAX point must be referenced on an axisymmetric harmonic element. If it is not, the program terminates the execution with an appropriate user fatal message. Further, the axisymmetric harmonic element (or elements) that reference GAX must have harmonic index 1 specified on the associated PAXSYMH entry. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. The G3D point must be on the axis of symmetry. The GAX point need not be on the axis of symmetry.
5. The G3D and GAX points must both have the same axial coordinate along the axis of symmetry.
6. The G3D point is regarded as the independent point and the GAX point is regarded as the dependent point. The dependent degrees of freedom of GAX will be selected from the axisymmetric harmonic degrees of freedom.

7. If the GAX point is on the axis of symmetry, the connection is made only to the G3D displacement components perpendicular to the axis. However, if the GAX point is not on the axis of symmetry, then connection is made not only to the G3D displacement components perpendicular to the axis, but also to the slopes. The tolerance to determine if the GAX point is on the axis of symmetry is 1.0E-6.

**RBE1****Rigid Body Element, Form 1**

Defines a rigid body connected to an arbitrary number of grid points.

**Format:**

| 1    | 2    | 3   | 4   | 5      | 6     | 7   | 8   | 9 | 10 |
|------|------|-----|-----|--------|-------|-----|-----|---|----|
| RBE1 | EID  | GN1 | CN1 | GN2    | CN2   | GN3 | CN3 |   |    |
|      |      | GN4 | CN4 | GN5    | CN5   | GN6 | CN6 |   |    |
|      | “UM” | GM1 | CM1 | GM2    | CM2   | GM3 | CM3 |   |    |
|      |      | GM4 | CM4 | -etc.- | ALPHA |     |     |   |    |

**Example:**

|      |    |    |        |       |  |  |  |  |  |
|------|----|----|--------|-------|--|--|--|--|--|
| RBE1 | 59 | 59 | 123456 |       |  |  |  |  |  |
|      | UM | 61 | 246    | 6.5-6 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| GNi        | Grid points at which independent degrees-of-freedom for the element are assigned. (Integer > 0)  |
| CNi        | Independent degrees-of-freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark 1. (Integers 1 through 6 with no embedded blanks.) |
| “UM”       | Indicates the start of the dependent degrees-of-freedom. (Character)   |
| GMj        | Grid points at which dependent degrees-of-freedom are assigned. (Integer > 0)  |
| CMj        | Dependent degrees-of-freedom in the global coordinate system at grid point(s) GMj. (Integers 1 through 6 with no embedded blanks.)                                       |
| ALPHA      | Thermal expansion coefficient. See Remark 13. (Real or blank)  |

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CMj.
- For the linear method, the total number of components in CN1 to CN6 must equal six; for example, CN1 = 123, CN2 = 3, CN3 = 2, CN4 = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. The first continuation entry is not required if there are fewer than four GN points. For the Lagrange method, the total number of components must also be six. In addition, CN1 must be 123456, and CN2 through CN6 must be blank.

4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding to m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
7. Element identification numbers should be unique with respect to all other element identification numbers.
8. RBE1, among other eligible rigid element types, can be selected via MPC and SET3.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
10. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
11. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
13. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN1 and any dependent grid point are taken as the average temperature of the two connected grid points.  
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE1.

**RBE2****Rigid Body Element, Form 2**

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.

**Format:**

| 1    | 2   | 3   | 4   | 5      | 6     | 7   | 8   | 9   | 10 |
|------|-----|-----|-----|--------|-------|-----|-----|-----|----|
| RBE2 | EID | GN  | CM  | GM1    | GM2   | GM3 | GM4 | GM5 |    |
|      | GM6 | GM7 | GM8 | -etc.- | ALPHA |     |     |     |    |

**Example:**

|      |    |       |    |    |    |    |    |    |  |
|------|----|-------|----|----|----|----|----|----|--|
| RBE2 | 9  | 8     | 12 | 10 | 12 | 14 | 15 | 16 |  |
|      | 20 | 6.5-6 |    |    |    |    |    |    |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| GN         | Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)  |
| CM         | Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.) See Remark 12. |
| GMi        | Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)  |
| ALPHA      | Thermal expansion coefficient. See Remark 11. (Real or blank)  |

**Remarks:**

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom which is obtained by CM multiplied with the number of dependent grid points.
3. For the linear method, the dependent degrees-of-freedom indicated by CM will be made members of the m-set at all grid points. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
4. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

5. Element identification numbers should be unique with respect to all other element identification numbers.
6. RBE2, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN and any dependent grid point are taken as the average temperature of the two connected grid points.  
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE2.
12. It can be any combination in 1, 2, 3, but must have all "456" if any rotation dof is used when there is large rotation.

**RBE2GS****Internally Generate an RBE2 Element**

Defines an RBE2 connecting the two closest grids to GS.

**Format:**

| 1      | 2   | 3  | 4    | 5   | 6   | 7 | 8  | 9     | 10 |
|--------|-----|----|------|-----|-----|---|----|-------|----|
| RBE2GS | EID | GS | TYPE |     |     | R | CM | ALPHA |    |
|        | XS  | YS | ZS   | GNi | GMj |   |    |       |    |

**Example:**

|        |   |    |  |               |    |     |  |       |  |
|--------|---|----|--|---------------|----|-----|--|-------|--|
| RBE2GS | 3 | 17 |  |               |    | 1.3 |  | 6.5-6 |  |
|        |   |    |  | end/ <i>l</i> | 56 | 99  |  |       |  |

**Alternate Formats and Examples:**

|        |     |      |              |     |      |     |               |       |  |
|--------|-----|------|--------------|-----|------|-----|---------------|-------|--|
| RBE2GS | EID | GS   | TYPE         |     |      | R   | CM            | ALPHA |  |
|        | XS  | YS   | ZS           | GNi | THRU | GNj | end/ <i>l</i> |       |  |
|        | GMk | THRU | GM/ <i>l</i> |     |      |     |               |       |  |

|        |    |    |  |    |      |      |               |    |  |
|--------|----|----|--|----|------|------|---------------|----|--|
| RBE2GS | 15 | 35 |  |    |      | -.66 |               |    |  |
|        |    |    |  | 88 | THRU | 107  | end/ <i>l</i> | 15 |  |
|        | 76 | 88 |  |    |      |      |               |    |  |

|        |       |      |         |    |    |     |               |  |  |
|--------|-------|------|---------|----|----|-----|---------------|--|--|
| RBE2GS | 25    |      |         |    |    | .66 |               |  |  |
|        | 5.173 | 0.0  | 19.3185 | 88 | 99 | 108 | end/ <i>l</i> |  |  |
|        | 15    | THRU | 88      |    |    |     |               |  |  |

|        |    |      |     |    |      |      |               |    |  |
|--------|----|------|-----|----|------|------|---------------|----|--|
| RBE2GS | 35 | 28   |     |    |      | -.66 |               |    |  |
|        |    |      |     | 56 | THRU | 102  | end/ <i>l</i> | 19 |  |
|        | 21 | THRU | 200 |    |      |      |               |    |  |

|        |      |      |      |               |      |      |      |  |  |
|--------|------|------|------|---------------|------|------|------|--|--|
| RBE2GS | 45   | 16   |      |               |      | -.66 |      |  |  |
|        |      |      |      | 56            | THRU | 102  | 1129 |  |  |
|        | 1146 | THRU | 1200 | end/ <i>l</i> |      |      |      |  |  |

**Descriptor****Meaning**

EID              Element identification number. (0 < Integer < 100,000,000)

GS              Search POINT or GRID point. (Integer > 0 or blank)

TYPE            Connectivity: (Character)

blank            Search the complete model. (Default)

| Descriptor | Meaning  |
|------------|--|
| NMFLIP     | The independent and dependent DOF's are interchanged.  |
| IIRBE2     | The grids chosen will be the independent GN's of the two closest existing RBE2 elements.   |
| NMIIRBE2   | Same as 'IIRBE2' except the independent and dependent grids are interchanged.  |
| R          | Radius. (Real > 0, or < 0)   |
| CM         | Component number of dependent degrees-of-freedom for grid GM. (Integers 1 through 6 with no embedded blanks. Blank defaults to 123456)   |
| ALPHA      | Thermal expansion coefficient. (Real or blank)   |
| XS, YS, ZS | Location of search point if GS is blank. Used only if GS=0 or blank. (Real or blank)   |
| GNi        | List of grids to be excluded from candidate grids for GN. If a GNi list is given it must end with an <i>endl</i> . (Integer > 0 or blank or "THRU")                                    |
| GMj        | List of grids to be excluded from candidate grids for GM. If no GNi list is given and a GMj list is present, then GNi must have an <i>endl</i> entry. (Integer > 0 or blank or "THRU") |

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. This entry will internally define a RBE2 element with the same ID of the RBE2GS entry. The grid assigned to GN will always be the independent grid. GM will be the dependent grid. If GS is a 'POINT' entry (or GS is blank and XYZ is specified), the first two grids that fall within the search radius  $|R|$  about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid GN the next closest will be the dependent grid GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj (and if TYPE = 'NMFLIP', then GN and GM will be reversed). If GS is 'GRID' entry, and is part of the physical model, i.e., it has physical structural elements attached to it, it will become GN the independent grid for the RBE2 to be generated. If it is in the GNi exclusion list, the next closest grid will be chosen. the closest grid within the search radius about  $|R|$  GN will be chosen as GM, the dependent grid. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with our without use of the exclusion lists for GNi and GMj) and if TYPE = 'NMFLIP', then GN and GM will be reversed. If GS is a 'GRID' entry, and is determined not to be part of the physical model, the first two grids that fall within the search radius  $|R|$  about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid, GN, the next closest will be the dependent grid, GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj) and if TYPE='NMFLIP', then GN and GM will be reversed. The GS grid will remain on the GEOM1 table for post-processing viewing purposes, but will not be part of the Nastran solution g-set. If TYPE='IIRBE2', the two grids chosen for the

RBE2 will be the independent GN's of existing RBE2 elements whose independent grids lie within the search radius. If TYPE='NMIIRB2', then GN and GM will be reversed. If no existing RBE2 elements have independent grids within the search radius or if only one existing RBE2 has an independent grid within the search radius, then a fatal message will be issued. POINT and GRID entries must be unique with respect to all other POINT and GRID entries. If TYPE='IIRB2', it is recommended that PARAM,AUTOMSET,YES be used in the analysis run.

3. If  $R < 0$ , the two located grids GN and GM will be made coincident to the GS (or XYZ) location.
4. If CM is 123456 and GM is touching only solid elements, CM will be internally changed to 123. For solid elements, the grids GN and GM should always be coincident.
5. When Module GP4 is run, checks are made to insure that the selected grids, GN and GM, do not violate existing constraint sets. If a violation occurs a fatal message will be issued for the offending grids. These grids can be excluded from further search inclusion by specifying them on the GNi or GMj list. PARAM,AUTOMSET,YES will often prevent any violation.
6. If GS=0 or blank and XS, YX, ZS is not specified or if both GS and XS, YS, ZS are specified a fatal error will occur.
7. The end of a grid exclusion lists is indicated by the existence of "end/" in the field following the last entry in the list. In the "thru" option, not all grids in the range need exist.
8. For superelement or part superelement connection the independent grid assigned to GN will be exterior to the superelement. The dependent grid GM will be an interior grid to the superelement. If the resulting RBE2 element, connects two different superelements, the element will be assigned to the superelement with the lower ID and the grid attached to the superelement with the higher ID will be moved to the superelement with the lower ID.
9. If the RBE2GS is listed on a SEELT entry, it will be placed as the SEELT directs. If say grid G1 lies in another superelement and G2 lies in the SEELT superelement, G1 will be moved to the SEELT defined superelement.
10. "THRU" should not be the first nonblank field of a continuation line. Blank fields are allowed in the exclusion lists for readability.
11. Use PARAM,SEP1XOVR,16 to print the grids found by each RBE2GS entry.
12. RBE2GS is not supported in the presence of part (BEGIN SUPER) superelements.

For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN1 and any dependent grid point are taken as the average temperature of the two connected grid points.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE2GS.

**RBE3**

## Interpolation Constraint Element

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

**Format:**

| 1    | 2       | 3     | 4       | 5    | 6    | 7      | 8    | 9      | 10 |
|------|---------|-------|---------|------|------|--------|------|--------|----|
| RBE3 | EID     |       | REFGRID | REFC | WT1  | C1     | G1,1 | G1,2   |    |
|      | G1,3    | WT2   | C2      | G2,1 | G2,2 | -etc.- | WT3  | C3     |    |
|      | G3,1    | G3,2  | -etc.-  | WT4  | C4   | G4,1   | G4,2 | -etc.- |    |
|      | "UM"    | GM1   | CM1     | GM2  | CM2  | GM3    | CM3  |        |    |
|      |         | GM4   | CM4     | GM5  | CM5  | -etc.- |      |        |    |
|      | "ALPHA" | ALPHA |         |      |      |        |      |        |    |

**Example:**

|      |       |       |     |      |     |     |     |   |  |
|------|-------|-------|-----|------|-----|-----|-----|---|--|
| RBE3 | 14    |       | 100 | 1234 | 1.0 | 123 | 1   | 3 |  |
|      | 5     | 4.7   | 1   | 2    | 4   | 6   | 5.2 | 2 |  |
|      | 7     | 8     | 9   | 5.1  | 1   | 15  | 16  |   |  |
|      | UM    | 100   | 14  | 5    | 3   | 7   | 2   |   |  |
|      | ALPHA | 6.5-6 |     |      |     |     |     |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. Unique with respect to all elements.<br>(0 < Integer < 100,000,000)  |
| REFGRID    | Reference grid point identification number. (Integer > 0)   |
| REFC       | Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)   |
| WTi        | Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)  |
| Ci         | Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)   |
| Gi,j       | Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)   |
| "UM"       | Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character) |
| GMi        | Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)   |

| Descriptor | Meaning  |
|------------|--|
| CMi        | Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.) |
| "ALPHA"    | Indicates that the next number is the coefficient of thermal expansion. (Character)                              |
| ALPHA      | Thermal expansion coefficient. See Remark 14. (Real or blank)  |

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, the REFC must be "123", "456", or "123456". No other combination is allowed.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of degrees-of-freedom given by REFC.
- For the linear method, the dependent degrees-of-freedom indicated by REFC will be made members of the m-set. For Lagrange rigid element, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
- We recommend that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.
- Blank spaces may be left at the end of a Gi,j sequence.
- For the Lagrange method, the default for "UM" must be used. For the linear method, the default for "UM" should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for "UM":
  - The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).
  - The components specified after "UM" must be a subset of the components specified under REFC and (Gi,j, Ci).
  - The m-rows by m-columns partition [Rm,m] of the global stiffness containing the dependent degrees-of-freedom of multipoint constraints and rigid elements must be nonsingular. PARAM,CHECKOUT in SOLs 101 through 200 may be used to check for this condition. For some complex constraints, forming a well-conditioned [Rm,m] is difficult for a user to perform manually and PARAM,AUTOMSET,YES will automatically determine a well-conditioned partition by employing a rectangular decomposition of the [Rm,g] matrix.
- Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
- RBE3, among other eligible rigid element types, can be selected via MPC and SET3.

10. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
11. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
13. The formulation for the RBE3 element was changed in Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees-of-freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark 5. The formulation prior to Version 70.7 may be obtained by setting SYSTEM(310)=1.
14. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the reference grid point REFGRID and any other grid point Gij are taken as the average temperature of the two connected grid points.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RBE3.

**RBE3U**

Defines Method to Distribute Applied Loads to a Surface in SOL 600

**Format:**

| 1     | 2   | 3    | 4    | 5  | 6    | 7  | 8  | 9  | 10 |
|-------|-----|------|------|----|------|----|----|----|----|
| RBE3U | EID | IOPT | GREF | G1 | G2   | G3 | G4 | G5 |    |
|       | G6  | G7   | G8   | G9 | etc. |    |    |    |    |

**Alternate Format:**

| 1     | 2   | 3    | 4    | 5  | 6    | 7    | 8  | 9  | 10 |
|-------|-----|------|------|----|------|------|----|----|----|
| RBE3U | EID | IOPT | GREF | G1 | THRU | G3   | BY | G4 |    |
|       | G5  | THRU | G6   | BY | G7   |      |    |    |    |
|       | G8  | THRU | G9   | BY | G10  | etc. |    |    |    |

**Examples:**

|       |     |   |      |     |      |     |  |  |  |
|-------|-----|---|------|-----|------|-----|--|--|--|
| RBE3U | 250 | 1 | 1000 | 100 | THRU | 120 |  |  |  |
|-------|-----|---|------|-----|------|-----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Identification number, unique among all elements. (Integer; no Default; Required)           |
| IOPT       | Option of how to distribute the load. (Integer; Default = 1)                                |
| 1          | Equal loads values will be applied to all grid points on the surface.                       |
| 2          | Loads will be distributed according to how close each grid point on the surface is to GREF. |
| Gi         | List of grids to which forces at GREF will be distributed. (Integer; Default = 123)         |

**Remarks:**

- The sum of the loads on all grid points is the same as the applied to GREF in each of the CREF directions. If IOPT=1 the force on each grid point will be the total loads divided by the number of applicable grid points. If IOPT=2 the load of each grid point will be weighted according to its distance from GREF, however the sum of all loads will be the same as that applied to GREF. The load is removed from GREF after distribution.
- RBE3U is only used to distributed applied loads. No actual rigid elements are included in the analysis. If rigid elements are desired, use RBE2 or RBE3.
- Only FORCE and MOMENT will be distributed using this entry. All other types of forces such as FORCE1, MOMENT1, TEMP, etc. applied to GREF will not be distributed.

**RBJOINT**

Defines a Joint Between Two Rigid Bodies

Defines a joint between two rigid bodies. Used in SOL 700 only.

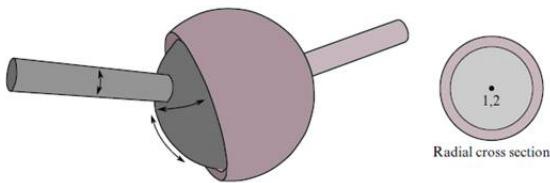
**Format:**

| 1       | 2  | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|------|----|----|----|----|----|----|----|
| RBJOINT | ID | TYPE | N1 | N2 | N3 | N4 | N5 | N6 |    |
|         |    | RPS  |    |    |    |    |    |    |    |

**Examples:****Example 1 - Cylindrical Joint**

|         |   |        |     |     |     |     |  |  |  |
|---------|---|--------|-----|-----|-----|-----|--|--|--|
| RBJOINT | 1 | CYLIND | 101 | 201 | 103 | 204 |  |  |  |
|---------|---|--------|-----|-----|-----|-----|--|--|--|

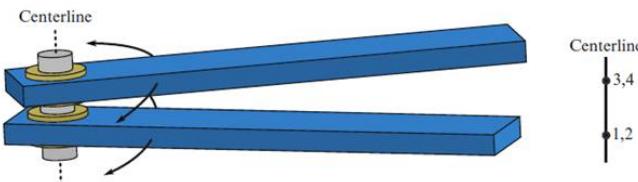
| Descriptor | Meaning  |
|------------|--|
| ID         | RBJOIN identification number. (Integer > 0; Required)  |
| TYPE       | Type of RBJOIN. (Character; Required)<br>Types available are (node numbers in the images correspond to N1, N2, etc.):<br>TYPE=SPHER(spherical), The relative motion of the rigid bodies are constrained so that nodes which are initially coincident remain coincident. In the below figure the socket's node is not interior to the socket-SOL700 does not require that a rigid body's nodes be interior to the body. |



Spherical joint

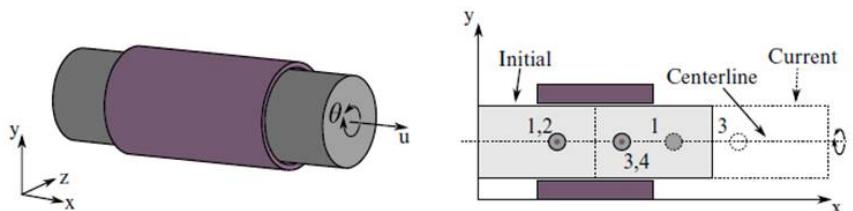
| Descriptor | Meaning |
|------------|---------|
|------------|---------|

TYPE=REVOLUTE, Both nodal pairs (1,2) and (3,4) are constrained to remain coincident. Consequently, the relative motion of these rigid bodies is restricted to rotations about the line segment formed by the two pairs of coincident nodes. This segment is labeled the "centerline".



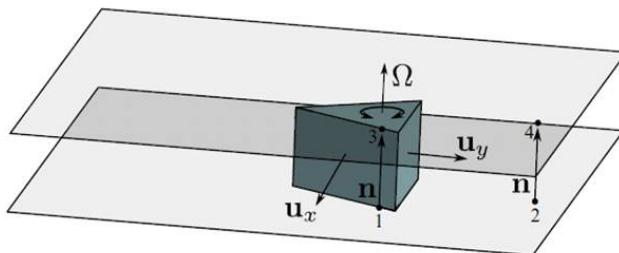
Revolute joint

TYPE=CYLINDRICAL, This joint is derived from the rotational joint by relaxing the constraints along the centerline. This joint admits relative rotation and translation along the centerline.



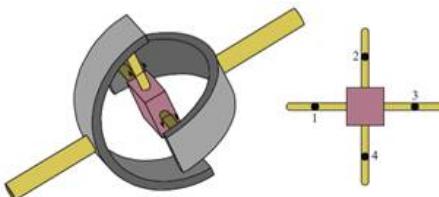
Cylindrical joint

TYPE=PLANAR, This joint is derived from the rotational joint by relaxing the constraints normal to the centerline. Relatively displacements along the direction of the centerline are excluded.



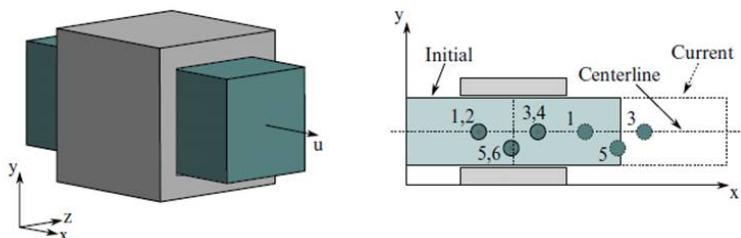
Planar joint

| Descriptor | Meaning   |
|------------|---|
|            | TYPE=UNIVERS (universal joint), In contrast with the preceding joints, nodal pairs (1,2) and (3,4) are not initially coincident. Rather, the segments formed by (1,3) and (2,4) must be orthogonal; and they serve as axes about which the two bodies may undergo relative rotation. The universal joint excludes all other relative motion and the axes remain orthogonal at all time. |



Universal joint

TYPE=TRANSL (translational), This is a cylindrical joint with a third pair of off-centerline nodes which restrict rotation. Aside from translation along the centerline the two rigid bodies are stuck together.



Translational joint

- N1 GRID id of Node 1, in rigid body A. Define for all joint types
- N2 GRID id of Node 2, in rigid body B. Define for all joint types
- N3 GRID id of Node 3, in rigid body A. Define for all joint types except SPHER.
- N4 GRID id of Node 4, in rigid body B. Define for all joint types except SPHER.
- N5 GRID id of Node 5, in rigid body A. Define only for joint types TRANSL.
- N6 GRID id of Node 6, in rigid body B. Define only for joint types TRANSL.
- RPS Relative penalty stiffness (Default = 1.0)

**RCONN****Rigid Connection**

Defines a rigid connection between the different parts of Lagrangian meshes (tied surfaces). Used in SOL 700 only.

**Format:**

| 1     | 2      | 3      | 4       | 5   | 6   | 7      | 8 | 9 | 10 |
|-------|--------|--------|---------|-----|-----|--------|---|---|----|
| RCONN | CID    | STYPE  | MTYPE   | SID | MID | OPTION |   |   |    |
|       |        |        |         |     |     |        |   |   |    |
|       | CLSGAP | GAPDIS | GAPDISV |     |     |        |   |   |    |

**Example:**

|       |   |      |      |   |   |        |  |  |  |
|-------|---|------|------|---|---|--------|--|--|--|
| RCONN | 7 | GRID | SURF | 3 | 7 | NORMAL |  |  |  |
|-------|---|------|------|---|---|--------|--|--|--|

| Field  | Content  |  |
|--------|--|--|
| CID    | Unique rigid-connection number (Integer; Required)   |  |
| STYPE  | Type of entity used to define the slave surface (Character; Default = SURF)  |  |
|        | SURF   | The faces of the elements are used for the slave surface. SID refers to BSURF ID. See Remark 1.  |
|        | GRID   | Grid points will be tied to the master surface. SID then refers to a SET or BCGRID ID containing the list of grid points to be used. See Remarks 2. and 3. |
| MTYPE  | Type of entity used to define the master surface (Character; Default = SURF)                                       |  |
|        | SURF   | The faces of the elements are used for the master surface. MID refers to BSURF ID.   |
| SID    | Slave BSURF ID or SET1 ID containing the list of grid points (Integer; Required)                                   |  |
| MID    | Master BSURF ID (Integer; Required)  |  |
| OPTION | Only used if discrete grid points are tied to a surface (STYPE is equal to GRID). (Character; Default = NORMAL)    |  |
|        | NORMAL   | The grid points are tied to the master surface. See Remark 2.  |
|        | SHELL  | The grid points are attached to the edge of shell or beam elements, which are tied to the shell surface. See Remark 3.                                     |
| CLSGAP | Switch to automatically close any gaps that are present between the master-slave surface (Character; Default = NO) |  |
|        | YES  | Gaps are automatically closed  |
|        | NO   | Gaps are not closed. See Remark 2.   |

| Field   | Content  |
|---------|--|
| GAPDIS  | Defines the tolerance used in the search for a master face. If the distance between a slave point and a master face falls within this tolerance, the master face is accepted. If not, the search for a correct master face continues (Character; Default = DISTANCE)<br><br>DISTANCE      The tolerance has the length as specified at GAPDISV |
| GAPDISV | The value of the gap tolerance or a factor to calculate this tolerance depending on the value of GAPDIS (Real; Default = 1.0E20)   |

## Remarks

1. The RCONN entry can be used to define three types of connection:
  - a. Two Surfaces Tied Together.
  - b. Define slave and master segments representing the two surfaces to be tied together. There should not be a gap between the two sets of segments. The two surfaces are tied together during the analysis.
  - c. Grid Points Tied to a Surface.
  - d. If STYPE is set to GRID and OPTION is set to NORMAL, the slave entities comprise discrete grid points that are tied to the master surface during the analysis. The grid points must lie on the surface.
  - e. Shell Edge Tied to a Shell Surface.
  - f. If STYPE is set to GRID and OPTION is set to SHELL, the edges of shell or beams elements can be tied to the faces of other shells. The grid points attached to the edge of the shells/beams must be selected as the slave grid points. The shell surface to which they are tied must be selected as the master surface. The two sets will then be tied together throughout the analysis. All degrees of freedom will be coupled.
2. The CLSGAP entry enables you to define two different meshes that are not coincident over the master/slave interface. If the option is set to YES, the slave surface becomes coincident (according to projections) with the master surface.
3. The search method of the contact algorithm is used to find the closest master face. The tolerance defined with the GAPDIS/GAPDISV fields is similar to the monitoring distance defined on the CONTACT entry with the MONDIS/ MONDISV fields.
4. The use of the gap closing CLSGAP can cause an element to collapse. This may happen if the GAPDISV tolerance is set to a value greater than the length of the side of an element.
5. When a solid and a shell mesh are tied together, the rotational degrees of freedom of the shell grid points are not coupled.
6. When OPTION=SHELL and CLSGAP=NO, the time step scale factor will be set to 0.4. This can be overwritten by:  
  
PARAM\*, STEPFCTRCONN, xxx

**RCPARM**

## Parameters for RC Heat Transfer Analysis Control

Defines a set of parameters for nonlinear resistance-capacitance based analysis.

**Format:**

| 1      | 2      | 3      | 4      | 5      | 6      | 7     | 8     | 9      | 10 |
|--------|--------|--------|--------|--------|--------|-------|-------|--------|----|
| RCPARM | ID     | SOLVER | DRLXCA | ARLXCA | BALENG | NLOOP | DAMPD | GRVCON |    |
|        | TIMEND | OUTPUT | CSGFAC | DTIMEI |        |       |       |        |    |

**Example:**

|        |       |        |       |       |       |      |     |      |  |
|--------|-------|--------|-------|-------|-------|------|-----|------|--|
| RCPARM | 1     | SNDUFR | 1.0-2 | 1.0-2 | 1.0-4 | 2000 | 0.7 | 9.81 |  |
|        | 1800. | 60.    | 1.2   | 0.0   |       |      |     |      |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number. See Remark 2. (Integer > 0; Required)  |
| Solver     | Equivalent Sinda Solution name. See Remarks 3. and 7. (String; Required) Possible values:<br>RCNS            “SNSOR”, “STDSTL”<br>RCNT            “SNFRDL”, “FWDBKL”, “SNADE”, “SNDUFR”, “ATSDUF”,<br>“ATSFBK”, “SNTSM”, “SNTSM3”, “SNTSM1”, “SNTSM4”,<br>Maintained for    “SNSOR1”, “SNSORA”, “SNSOR1A”, “SNHOSD”, “SNSOSS”,<br>legacy, but not    “SNHOSS”, “SCROUT”, “SNDSNR”, “TRSPMA”<br>shown in GUI |
| DRLXCA     | Diffusion node convergence criterion (Real $\geq 0.0$ ; Default 1.0e-3 degrees) (SimX:<br>Default 1.0e-4 for transient)   |
| ARLXCA     | Arithmetic node convergence criterion (Real $\geq 0.0$ ; Default 1.0e-3 degrees) (SimX:<br>Default 1.0e-4 for transient)  |
| BALENG     | Allowable system energy imbalance (Real $\geq 0.0$ ; Default 0.0 energy/time)   |
| NLOOP      | Number of iterations allowable (Integer $\geq 0$ ; Default 5000 loop) (SimX: Default 50 for<br>transient)   |
| DAMPD      | Damping constant (Real $\geq 0.0$ ; Default 0.0 non dimensional)  |
| GRVCON     | Gravitation constant (Real $\geq 0.0$ ; Default 9.81 length/time^2)   |
| TIMEND     | Problem end time (Real; Default 3600.0 time)  |
| OUTPUT     | Output interval (Real $\geq 0.0$ ; Default 60.0 time)   |
| CSGFAC     | Time step control factor (Real $\geq 0.0$ ; Default 1.0 non dimensional)  |
| DTIMEI     | Time step (Real $\geq 0.0$ ; Default 0.0 time)  |

**Remarks:**

1. This entry is for RC Network solver only.
2. ID can be referenced by the case control command. Different cases may reference different RCPARM entries. It is possible to have multiple RCPARM entries inside one BDF file.
3. RCNS stands for RC Network Steady-state (static), RCNT stands for RC Network Transient
4. The default values are for Nastran only. The modeler (Patran or SimXpert) will have their default values. Some are related with the model units. The default values will not be all 0.0.
5. Default solvers and result checking SNSOR and SNDUFR are the default steady and transient solvers. They are good for most of the cases. Users are encouraged to check the results by the following methods.

Obtain nearly identical results with a different solver

Obtain nearly identical results with tighter convergence

Obtain nearly identical results with a smaller time step (transient)

Obtain nearly identical results with more rays or different random seeds (external radiation solvers)

6. About non-default solvers:

For non-spacecraft, non-radiation problems:

Solid type model – SNSOR, SNTSM1

For spacecraft/radiation models:

SNSOR (with user-specified DAMPD if necessary)

FWDBKL if thermo-stats are present

ATSDUF, SNTSM1 for most other cases

7. About the time step

- a. The default computed time step (DTIMEU) = CSGMIN\* CSGFAC. CSGMIN can be checked in the .sot file. If CSGFAC is not specified, it is internally set to 1.0. The DTIMEU in the .sot file has been truncated by the OUTPUT point.
- b. In a normal sized model, CSGMIN usually is small enough for the time step which will assure a convergent transient run.
- c. CSGFAC is used to adjust the time step. It is recommended to determine the best CSGFAC to the model while maintaining acceptable temperature errors.
- d. If OUTPUT < CSGFAC\*CSGMIN or OUTPUT < DTIMEI, then OUTPUT becomes the time step. All the OUTPUT points are automatically required to be calculated.
- e. DTIMEI is the forced time step which will ignore any other factors. Sometimes it may lead to inaccurate answer if it is too large. DTIMEI does not affect the automatic time step solvers.
- f. If the model size is very small, CSGMIN may be too big for the time step. A small CSGFAC or DTIMEI should be used to adjust the time step.
- g. CSGFAC\*CSGMIN or DTIMEI should be small enough to “catch” any details in time fields, temperature fields or orbital flux arrays.

8. For more details about these parameters, please reference *MSC SINDA User's Guide and Library Reference*.

**RCROSS****Cross-Power Spectral Density and Cross-Correlation Functions Output**

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.

**Format:**

| 1      | 2   | 3      | 4   | 5     | 6      | 7   | 8     | 9     | 10 |
|--------|-----|--------|-----|-------|--------|-----|-------|-------|----|
| RCROSS | SID | RTYPE1 | ID1 | COMP1 | RTYPE2 | ID2 | COMP2 | CURID |    |

**Example:**

|        |    |      |     |   |        |     |    |   |  |
|--------|----|------|-----|---|--------|-----|----|---|--|
| RCROSS | 10 | DISP | 100 | 3 | STRESS | 200 | 10 | 2 |  |
|--------|----|------|-----|---|--------|-----|----|---|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Case Control RCROSS identification number for cross-power spectral density function and cross-correlation function. (Integer > 0) |
| RTYPEi     | Type of response quantity. At least one field must be selected. See Remark 2. (Character or blank)                                |
| IDi        | Element, grid or scalar point identification number. (Integer > 0)  |
| COMPi      | Component code (item) identification number. See Remark 3. (Integer > 0)  |
| CURID      | Curve identification number. See Remark 4. (Integer $\geq 0$ or blank)  |

**Remarks:**

1. This entry is required for computing the cross-power spectral density function and cross-correlation function. SID must be selected with the Case Control command (RCROSS = SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity, and fields RTYPE2, ID2, and COMP2 the second in the correlation.
2. The keywords for field RTYPEi are listed as follows:

| Keyword | Meaning                              |
|---------|--------------------------------------|
| DISP    | Displacement Vector                  |
| VELO    | Velocity Vector                      |
| ACCEL   | Acceleration Vector                  |
| OLOAD   | Applied Load Vector                  |
| SPCF    | Single-point Constraint Force Vector |

| Keyword | Meaning                             |
|---------|-------------------------------------|
| MPCF    | Multi-point Constraint Force Vector |
| STRESS  | Element Stress                      |
| STRAIN  | Element Strain                      |
| FORCE   | Element Force                       |

If anyone of RTYPE1 or RTYPE2 is blank, then the default is the one same as the other field.

3. For elements, the item code COMPi represents a component of the element stress, strain, and force and is described in Tables [Element Stress-Strain Item Codes, 1048](#) and [Element Force Item Codes Part 1, 1085](#). For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function. For grid point, the item code is one of 1, 2, 3, 4, 5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar point, always use 1.
4. Field CURID is optional. It is for the user's convenience to identify the output by using a single index.

**RELEASE****Superelement Boundary Grid Point Release**

Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

**Format:**

| 1       | 2    | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|--------|----|----|----|----|----|----|
| RELEASE | SEID | C  | G1     | G2 | G3 | G4 | G5 | G6 |    |
|         | G7   | G8 | -etc.- |    |    |    |    |    |    |

**Example:**

|         |     |     |   |   |    |   |     |   |  |
|---------|-----|-----|---|---|----|---|-----|---|--|
| RELEASE | 15  | 456 | 3 | 7 | 11 | 2 | 156 | 9 |  |
|         | 152 | 162 |   |   |    |   |     |   |  |

**Alternate Formats and Examples:**

|         |      |   |    |        |     |  |  |  |  |
|---------|------|---|----|--------|-----|--|--|--|--|
| RELEASE | SEID | C | G1 | “THRU” | G2  |  |  |  |  |
| RELEASE | 6    | 2 | 15 | THRU   | 127 |  |  |  |  |

(ALL must be in FIELD 4 and no continuation is allowed)

|         |      |     |       |  |  |  |  |  |  |
|---------|------|-----|-------|--|--|--|--|--|--|
| RELEASE | SEID | C   | “ALL” |  |  |  |  |  |  |
| RELEASE | 127  | 156 | ALL   |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. (Integer > 0)   |
| C          | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks.) |
| Gi         | Grid point identification numbers. (Integer > 0; “THRU”, or “ALL”; For THRU option, G1 < G2.)   |

**Remarks:**

1. A grid point referenced on this entry must be an exterior grid point of the superelement referenced on the entry.
2. In the first alternate format, all grid points in the sequence G1 through G2 are not required to be exterior grid points. Such grid points will collectively produce a warning message but will otherwise be ignored.
3. If the “ALL” option is used, all exterior grid points are released for the referenced degrees-of-freedom.
4. The RELEASE entry is applicable to only the superelement solution sequences (SOLs 101 through 200). It may not reference the residual structure (SEID = 0).
5. This entry is not supported for partitioned superelements.

**RELEX**

## External Definition of a Rigid Ellipsoid- SOL 700

Defines a rigid ellipsoid whose properties and motion are defined by either ATB. Used in SOL700 only.

**Format:**

| 1     | 2    | 3    | 4 | 5 | 6 | 7 | 8   | 9 | 10 |
|-------|------|------|---|---|---|---|-----|---|----|
| RELEX | NAME | PROG |   |   |   |   | etc |   |    |

**Example:**

|       |      |     |  |  |  |  |  |  |  |
|-------|------|-----|--|--|--|--|--|--|--|
| RELEX | HEAD | ATB |  |  |  |  |  |  |  |
|-------|------|-----|--|--|--|--|--|--|--|

| Field | Content   |
|-------|---|
| NAME  | This name is used within the input file to define the interactions between the external ellipsoid and grid points and rigid bodies. This name is also used in the output requests.<br><br>When coupled to ATB, the name must correspond to the name of the ATB segment. (Character; required) |
| PROG  | Name of the external program. (Character; required)<br><br>ATB SOL700 runs coupled with ATB   |

**RESTART**

## Restart Data for Marc Executed from SOL 600

Specifies writing or reading of restart data for Nonlinear Analysis when Marc is executed from SOL 600. If this Bulk Data entry is found in the job stream, the type of "restart" specified by KIND and KTYPE will be performed. Only one RESTART entry is allowed.

**Format:**

| 1       | 2    | 3    | 4     | 5       | 6      | 7     | 8       | 9       | 10 |
|---------|------|------|-------|---------|--------|-------|---------|---------|----|
| RESTART | ID   | KIND | NINC  | NBEGIN  |        |       |         |         |    |
|         | NAME |      | TSTEP | ENDTIME | NSTEPS | NDCYC | STEPMAX | PERCENT |    |

**Example(s):**

|         |     |   |   |  |  |  |  |  |  |
|---------|-----|---|---|--|--|--|--|--|--|
| RESTART | 101 | 1 | 1 |  |  |  |  |  |  |
|---------|-----|---|---|--|--|--|--|--|--|

The above example writes a restart file. The original run named is abcde.dat.

|         |       |   |   |    |  |  |  |  |  |
|---------|-------|---|---|----|--|--|--|--|--|
| RESTART | 201   | 3 | 1 | 15 |  |  |  |  |  |
|         | abcde |   |   |    |  |  |  |  |  |

The above example restarts the original abcde.dat run. The name of the restart run must not be abcde.dat in this case.

|         |          |      |  |    |  |  |  |  |  |
|---------|----------|------|--|----|--|--|--|--|--|
| RESTART | 151      | 2    |  | 11 |  |  |  |  |  |
|         | my_first | _run |  |    |  |  |  |  |  |

The above example reads the restart file and prints out results not printed in the original run.

| Descriptor      | Meaning   |
|-----------------|---|
| ID              | Identification number of the restart entry -- Not presently used (Integer)  |
| KIND<br>(2,1)   | Type of restart (Integer > 0; required field; no Default)<br>1 = Write a restart file<br>2 = Restart a previous analysis (Read an existing restart file)<br>3 = Restart a previous analysis and write new data on restart file.<br>11 = Only write restart file for the last converged increment of the run.<br>12 = Read a restart file written with KIND=11<br>13 = Read a restart file written with KIND=11 and write the last increment or time step of the present run on that file as well. |
| NBEGIN<br>(2,3) | The "time" increment at which the restart run begins (used only if KIND=2 or 3).<br>(Integer > 0. See Remarks 3, 4.)  |
| NINC<br>(2,2)   | Number of increments between writing of restart data. (Integer > 0; Default = 1)  |

| Descriptor       | Meaning   |
|------------------|---|
| IPRINT<br>(2,8)  | Set this field to 1 if the restart data is to be printed (All data from increment INCBEG to LAST will be printed if IPRINT=1). Use this option if printing on a previous run was suppressed but now is desired. (Integer $\geq 0$ ; Default = 0)  |
| LAST<br>(2,9)    | The last time increment is printed if IPRINT=1, otherwise this field is ignored. (Integer $> 0$ , Default = 0. See Remarks 3., 4.)  |
| NAME             | Name of input file for the original MSC Nastran run without extension. NAME is limited to 16 characters and may not contain imbedded blanks. If the small field format is used, NAME may span fields 2 and 3 of the continuation entry. If the large field is used, NAME should be coded in the 2nd field. NAME is required for a restart run, see Remarks 3., 7. If NAME exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large field fixed or free-format. |
| TSTEP<br>[2,1]   | Time step size after restart -- For dynamic problems only. (Real $\geq 0.0$ or blank; if the value is $\leq 0.0$ the original step size is used)  |
| ENDTIME<br>[2,2] | Ending time for this restart run (Real $\geq 0.0$ or blank, if the value is $\leq 0.0$ the original end time is used)   |
| NSTEPS<br>[2,3]  | Total number of time steps for the restart run plus the original run. (Integer $\geq 1$ ; Default = 1)  |
| NDCYC<br>[2,6]   | Desired number of recycles if AUTO INCREMENT options were specified on the original run. (Integer $\geq 0$ ; if the value is $\leq 0$ the original is used)   |
| STEPMAX<br>[2,7] | Maximum step size if AUTO INCREMENT options were specified on the original run. (Real $\geq 0.0$ ; if the value is $\leq 0$ the original is used)   |
| PERCENT<br>[2,8] | Percentage of total load to be applied.(Real $\geq 0.0$ ; if the value is $\leq 0$ the original is used)  |

**Remarks:**

1. RESTART is available only when Marc is executed from within MSC Nastran Implicit Nonlinear (SOL 600).
2. There should only be one RESTART entry in the bulk data. If more than one exists, the first one will be used.
3. (i,j) Indicate the field in Marc's RESTART model definition options. [i,j] Indicate the field in Marc's REAUTO model definition options.
4. The jid.marc.t16 and jid.marc.t08 files must be saved from the first run when a restart run is executed. Both original and restart runs must be located in the same directory. File extensions are .t08 and t16.
5. A restart run may not have the same input file name (jid) as that of the original run. In other words, if the input file for the original run is named abcd.dat, the input file for a restart run may not be named abcd.dat.

6. For static analyses, normally each load case has a total time of 1.0. The first case goes from 0.0 to 1.0, the second from 1.0 to 2.0, etc. If the first run has two static load cases and terminates at 1.6, it is in the middle of the second load case. The original run should be examined to determine which increment (before 1.6) to begin the restart run.
7. In the second example above, the original run was named my\_first\_run.dat (or first\_run.bdf, etc). This is a small field example. There are 8 characters in the 2nd field of the continuation line and 4 characters in the third field. The name can start anywhere within the two fields. There must not be any blank spaces in the name. If the name exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large-field fixed or free format. NAME is limited to a maximum of 16 characters.
8. Both original and restart run names (jid's) should use only lower case letters except on computer systems that are not case sensitive. "NAME" (continue line, fields 2-3) will be converted automatically to lower case.
9. The restart run must use the same values of Marc's DIST LOAD parameter as the original run or it may fail. Be sure to use parameters MARCDIS2, MARCDIS3 and MARCDIS4 to set these values to be exactly the same as the original run (examine jid.marc.dat from the original run to determine these values before submitting the new run).
10. The RSF options creates a complete database which is necessary for restarts. When RSF=0 is specified, the same file is overwritten after each interval. When RSF=1 is specified, a new restart file is created after each interval, thus a "family" of files is created numbered sequentially jid.dytr.d3dump10, jid.dytr.d3dump02, etc. These files can take significant disk space but are important if a model might need to be modified prior to the end time.
11. It is not necessary to enter continuation lines if not needed for the particular job to be run.
12. Restarts are not available for models with bolt loading using entries MBOLT, MBOLTUS or BOLT.

**RFORCE**

## Rotational Force

Defines a static loading condition due to an angular velocity and/or acceleration.

**Format:**

| 1      | 2    | 3  | 4    | 5 | 6  | 7  | 8  | 9      | 10 |
|--------|------|----|------|---|----|----|----|--------|----|
| RFORCE | SID  | G  | CID  | A | R1 | R2 | R3 | METHOD |    |
|        | RACC | MB | IDRF |   |    |    |    |        |    |

**Example:**

|        |     |   |  |      |     |     |     |   |  |
|--------|-----|---|--|------|-----|-----|-----|---|--|
| RFORCE | 2   | 5 |  | -6.4 | 0.0 | 0.0 | 1.0 | 2 |  |
|        | 1.0 |   |  |      |     |     |     |   |  |

| Descriptor          | Meaning  |
|---------------------|--|
| SID                 | Load set identification number. (Integer > 0)  |
| G                   | Grid point identification number through which the rotation vector acts. (Integer $\geq 0$ )   |
| CID                 | Coordinate system defining the components of the rotation vector. See Remark 16. (Integer $\geq 0$ ; Default = 0)  |
| A                   | Scale factor of the angular velocity in revolutions per unit time. (Real)  |
| R1, R2, R3          | Rectangular components of rotation vector $\vec{R}$ . The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$ unless A and RACC are both zero)   |
| METHOD              | Method used to compute centrifugal forces due to angular velocity, see Remark 20.. For angular acceleration, see Remark 13. (Integer = 1 or 2; Default = 1)  |
| RACC                | Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)   |
| MB                  | Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 15. (Integer; Default = 0)   |
| IDRF (SOL 600 only) | ID indicating to which portion of the structure this particular RFORCE entry applies. It is possible to have multiple RFORCE entries in the same subcase for SOL 600 to represent different portions of the structure with different rotational accelerations. IDRF corresponds to a SET3 entry specifying the elements with this acceleration. A BRKSQ entry may also be specified with a matching IDRF entry. (Integer; Default = 0) |

**Remarks:**

- The forces that are created with the RFORCE entry act on the structure as follows: the forces that are defined with the RFORCE entry for a constant angular velocity (A), act in the positive radial direction. These forces represent the inertia forces on the structure due to a constant angular velocity. The forces that are defined with the RFORCE entry for a constant angular acceleration (RACC), act in the same direction as the angular acceleration. These forces would be opposite to the inertia forces on the structure due to a constant angular acceleration. In [Figure 9-151](#), the force vector at grid point Gi is given by

$$\{\vec{F}\}_i = [m]_i [\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)] \quad (9-33)$$

where:

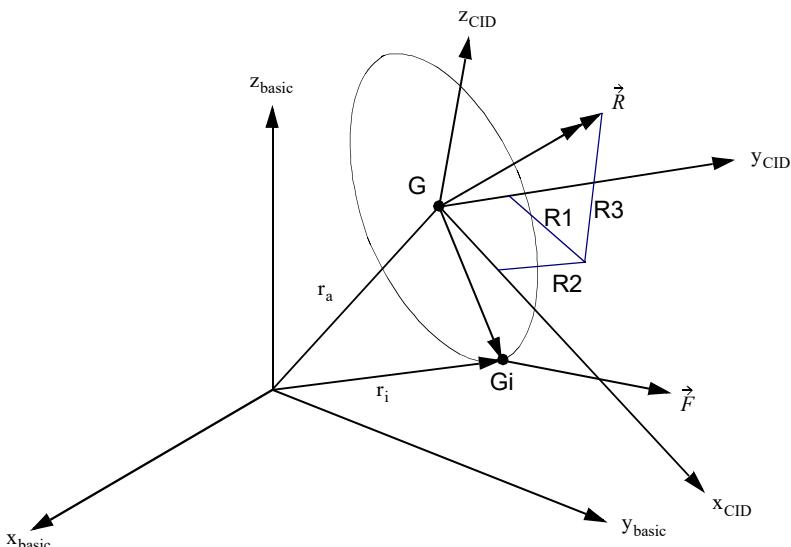
$$\text{angular velocity} = \vec{\omega} = 2\pi A \cdot \vec{R} \text{ (radians/unit time)}$$

$$\text{angular acceleration} = \vec{\alpha} = 2\pi RACC \cdot \vec{R} \text{ (radians/unit time}^2\text{)}$$

$$[m]_i = 3 \times 3 \text{ translational mass matrix at grid point Gi}$$

**Note:**

The equation for  $\vec{F}_i$  will have additional terms if the mass is offset or  $I_{23}^i$ ,  $I_{13}^i$  terms exist relative to the rotation axes and METHOD = 1 is selected.



[Figure 9-151](#) RFORCE Vector at Grid Point Gi

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. G = 0 signifies that the rotation vector acts through the origin of the basic coordinate system.
4. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.
5. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.
6. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

|         | No Offset                | Offset     |
|---------|--------------------------|------------|
| Lumped  | METHOD = 1 or METHOD = 2 | METHOD = 1 |
| Coupled | METHOD = 2               | Neither    |

7. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.
8. For superelement analysis, G should reference a residual structure point that is exterior to all superelements when loading Superelement 0 and a separate G must be interior to each superelement when loading an upstream superelement else centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.
9. In a geometric nonlinear static analysis (SOL 106 and 400 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.
10. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE input (and not the angular velocity vector) is scaled linearly. This means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE loading is applied in a single increment.
11. The continuation entry is optional.
12. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.
13. Loads derived from this entry do not include effects due to mass specified for scalar points.

14. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter **FOLLOWK** (Ch. 6)). In addition, follower force effects are included in the force balance in the nonlinear solution sequences, SOLs 106, 129, 153, 159 and 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) and general nonlinear solution sequence (SOL 400) for both ANALYSIS=NSTAT and NLTRAN but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
15. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
16. If CID is not a rectangular coordinate system, RFORCE will treat it as if it were and unexpected answers may result.
17. Follower force stiffness (param,followk,yes) is supported for method 2 only.
18. Multiple RFORCE entries with different SID's may be used in SOL 600 in the same subcase or SOL 400 in the same loadcase (STEP and/or SUBCASE). They should be combined using the LOAD entry similar to the way FORCE or PLOAD4 with different ID's are combined.
19. Fields CID, METHOD, RACC, MB and IDR will be ignored for SOL 700.
20. For Axisymmetric Harmonic elements, (elements selected with PAXSYMH entry), METHOD=2 is required. A selection of METHOD=1 will automatically be overridden and be replaced by METHOD=2 for all elements of the model.
21. When used with rotor dynamics, the angular velocity creates axial terms that are proportional to  $\omega^2$  and moments that are proportional to  $\omega$ . In order to keep consistent spin rate, if the scaling is requested on the LOAD entry, the on the axial terms will be scaled by this scale factor, and the  $\omega$  on the moment terms will be scaled by the square root of this scale factor. See Remark 6. of the LOAD entry.
22. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RGYRO**

## Rotordynamic Analysis Parameters

Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.

**Format:**

| 1     | 2      | 3       | 4       | 5       | 6       | 7       | 8     | 9        | 10 |
|-------|--------|---------|---------|---------|---------|---------|-------|----------|----|
| RGYRO | RID    | SYNCFLG | REFROTR | SPDUNIT | SPDLLOW | SPDHIGH | SPEED | ROTRSEID |    |
|       | WR3WRL | WR4WRL  | WRHWRL  |         |         |         |       |          |    |

**Example:**

| 1     | 2   | 3    | 4 | 5   | 6      | 7      | 8 | 9  | 10 |
|-------|-----|------|---|-----|--------|--------|---|----|----|
| RGYRO | 100 | SYNC | 1 | RPM | 1000.0 | 5000.0 |   | 10 |    |

| Descriptor | Meaning   |
|------------|---|
| RID        | Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required; no Default). See Remarks 1. and 2.  |
| SYNCFLG    | Specifies whether the analysis is synchronous or asynchronous analysis. Required input for frequency response and complex modes analyses. Not required for static analyses. (Character: 'SYNC', 'ASYNC', or blank). See Remarks 2. through 4. |
| REFROTR    | Specifies the reference rotor ID for the analysis. (Integer > 0; Required; no Default)  |
| SPDUNIT    | Specifies whether the entries SPDLLOW, SPDHIGH, and SPEED are given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/unit time). (Character: 'RPM' or 'FREQ'; no Default). See Remark 2.                               |
| SPDLLOW    | Specifies the low speed for synchronous analysis. See Remark 2. (Real; Default = 0.0)   |
| SPDHIGH    | Specifies the high speed for synchronous analysis. See Remark 2. (Real; Default = 99999.0)  |
| SPEED      | Specifies reference rotor speed for asynchronous analysis. Also required for static analyses. See Remark 2. (Default = 0)   |
| ROTRSEID   | Identification number of the superelement in which the rotor specified in the REFROTR field is defined. (Integer >= 0, Default = 0).  |
| WR3WRL     | Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field for asynchronous complex eigenvalue analysis. (Real, Default = 0.)                                   |
| WR4WRL     | Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries for asynchronous complex eigenvalue analysis. (Real, Default = 0.)                           |
| WRHWRL     | Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card for asynchronous complex eigenvalue analysis. (Real, Default = 0.)                        |

**Remarks:**

1. Multiple RGYRO entries with the same RID value are not allowed.
2. The required information on the RGYRO entries varies for different analyses. Values for the RID and SPDUNIT fields are always required. Values for SPDLOW, SPDHIGH and SPEED are analysis dependent as shown in the table below:

| Solution Sequence  | Type of Analysis | PARAM, GYROAVG | Required Entry  | COMMENT         |
|--------------------|------------------|----------------|-----------------|-----------------|
| Frequency Response | SYNC             | 0              | None            | --              |
|                    | SYNC             | -1             | SPDLOW, SPDHIGH | a, b            |
|                    | ASYNC            | 0              | SPEED           | --              |
|                    | ASYNC            | -1             | SPEED           | b               |
| Complex Modes      | SYNC             | --             | SPDLOW, SPDHIGH | a, b            |
|                    | ASYNC            | --             | SPEED           | b,<br>Remark 4. |
| Static Analysis    | --               | --             | SPEED           | Remark 3.       |

- a. The relative rotor speeds will be treated as linearly dependent on the reference rotor speed ( $\Omega = A0 + A1\Omega_{\text{reference}}$ ). The scale factors A0 and A1 will be determined by a least-mean-square fit of the relative rotor speeds input on the RSPINR entries between SPDLOW and SPDHIGH of the reference rotor. If SPDLOW or SPDHIGH are outside the range specified on the RSPINR entry, the values will be extrapolated from the RSPINR entry values.
- b. PARAM, WR3 and PARAM, WR4 are required for rotor damping.
3. For static analysis, the SYNCFLG field must be left blank or must have the ASYNC option specified in it. Otherwise, the program terminates the execution with an appropriate fatal message.
4. When there is a Case Control request for Campbell diagram, the selected RGYRO Bulk Data entry must have the ASYNC option specified in its SYNCFLG field. Otherwise, the program terminates the execution with an appropriate fatal message.
5. In the presence of GR field, the rotor damping matrix (BR) in complex eigenvalue analysis is given by:  

$$BR = KR1 \cdot GR1 / |\Omega_1(\Omega_{\text{ref}}) - WR3WRL|$$

where,

KR is the rotor stiffness matrix.

GR is the structural damping parameters specified in the RSPINR entry

$\Omega_1(\Omega_{\text{ref}})$  is the specified rotor speed.
6. If non-zero WR3WRL is specified in RGYRO, then PARAM,WR3 and WR3R (in RSPINR) cannot be specified. Appropriate FATAL message will be issued, in case these entries are specified.

**RINGAX**

## Conical Shell Ring

Defines a ring for conical shell problems.

**Format:**

| 1      | 2  | 3 | 4 | 5 | 6 | 7 | 8  | 9 | 10 |
|--------|----|---|---|---|---|---|----|---|----|
| RINGAX | ID |   | R | Z |   |   | PS |   |    |

**Example:**

|        |   |  |     |       |  |  |     |  |  |
|--------|---|--|-----|-------|--|--|-----|--|--|
| RINGAX | 3 |  | 2.0 | -10.0 |  |  | 162 |  |  |
|--------|---|--|-----|-------|--|--|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Ring identification number. See Remark 6. (Integer > 0)   |
| R          | Ring radius. (Real > 0.0)   |
| Z          | Ring axial location. (Real)   |
| PS         | Permanent single-point constraints. (Any unique combination of the Integers 1 through 6 with no embedded blanks.) |

**Remarks:**

1. RINGAX is allowed only if an AXIC entry is also present.
2. The number of degrees-of-freedom defined is  $(6 - \text{NPS}) \cdot H$  where  $H$  is the harmonic count and NPS is the number of digits in field 8. (See [AXIC, 1245](#)).
3. RINGAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.
5. Constraints may be necessary to avoid matrix singularities. The CONEAX element has no stiffness for rotation about the normal. In addition, there is no stiffness for rotation about V (see [Figure 9-152](#)) when transverse shear flexibility is not included.

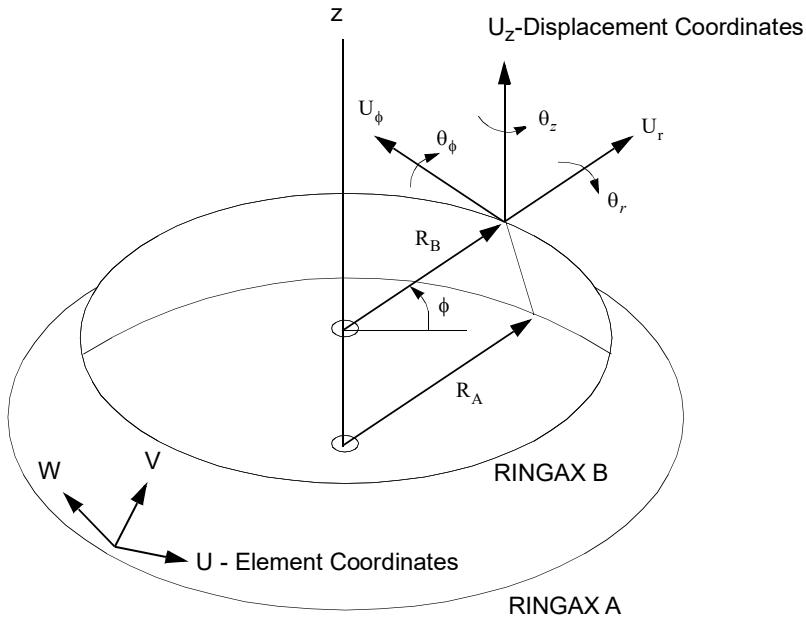


Figure 9-152 RINGAX Coordinate System

6. In order to reference this entry on a SET Case Control command, the ID must be modified by  $ID(n) = ID + 1000000 \cdot n$  where n is the harmonic number plus one and ID(n) is the value specified on the SET entry.

**RINGFL****Axisymmetric Fluid Point**

Defines a circle (fluid point) in an axisymmetric fluid model.

**Format:**

| 1      | 2    | 3   | 4   | 5   | 6    | 7   | 8   | 9   | 10 |
|--------|------|-----|-----|-----|------|-----|-----|-----|----|
| RINGFL | IDFA | XA1 | XA2 | XA3 | IDFB | XB1 | XB2 | XB3 |    |

**Example:**

|        |   |     |  |      |  |  |  |  |  |
|--------|---|-----|--|------|--|--|--|--|--|
| RINGFL | 3 | 1.0 |  | 30.0 |  |  |  |  |  |
|--------|---|-----|--|------|--|--|--|--|--|

| Descriptor                        | Meaning   |
|-----------------------------------|---|
| IDFA, IDFB                        | Unique identification number of the fluid points. (0 < Integer < 500000)  |
| XA <sub>i</sub> , XB <sub>i</sub> | Coordinates of the point defined in the coordinate system specified on the AXIF entry.<br>(Real; XA1 and XB1 > 0.0) |

**Remarks:**

1. RINGFL is allowed only if an AXIF entry is also present.
2. All fluid point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. X1, X2, X3 are (r,  $\phi$ , z) for a cylindrical coordinate system and ( $\rho$ ,  $\theta$ ,  $\phi$ ) for a spherical coordinate system.  $\theta$  is in degrees. The value of  $\phi$  must be blank or zero.
4. One or two fluid points may be defined per entry.

**RJOINT****Rigid Joint**

Defines a rigid joint element connecting two coinciding grid points.

**Format:**

| 1      | 2   | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|--------|-----|----|----|----|---|---|---|---|----|
| RJOINT | EID | GA | GB | CB |   |   |   |   |    |

**Example:**

|        |   |   |   |       |  |  |  |  |  |
|--------|---|---|---|-------|--|--|--|--|--|
| RJOINT | 5 | 1 | 2 | 12345 |  |  |  |  |  |
|--------|---|---|---|-------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Element identification number. (Integer > 0)  |
| GA, GB     | Grid point identification numbers. (Integer > 0)  |
| CB         | Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. (Integers 1 through 6 with no embedded or blank.) |

**Remarks:**

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
3. The length between grid points GA and GB must be zero.
4. When CB = "123456" or blank, the grid point GB is constrained to move with GA and the two grid points moves as a single point. For default CB = "123456".
5. If any degree-of-freedom is released on CB, RJOINT becomes a mechanical joint element. For example, CB = "12345", then RJOINT becomes a hinge. CB = "1234", then RJOINT becomes a universal joint. And CB = "123", RJOINT becomes a spherical joint.
6. For the Lagrange method, the theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB.
7. Thermal expansion effect is not applicable for the RJOINT element, since the distance between grid points GA and GB is zero.
8. Element identification numbers should be unique with respect to all other element identification numbers.

**RLOAD1**

## Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form

$$\{P(f)\} = \{A\}[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}$$

for use in frequency response problems.

**Format:**

| 1      | 2   | 3        | 4                 | 5                   | 6     | 7     | 8    | 9 | 10 |
|--------|-----|----------|-------------------|---------------------|-------|-------|------|---|----|
| RLOAD1 | SID | EXCITEID | DELAYI/<br>DELAYR | DPHASEI/<br>DPHASER | TC/RC | TD/RD | TYPE |   |    |

**Example:**

|        |   |   |     |    |   |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|
| RLOAD1 | 5 | 3 | 2.0 | 10 | 1 |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. See Remarks 1. and 3. (Integer > 0)  |
| EXCITEID   | Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$ . See Remarks 4. and 5. (Integer > 0) |
| DELAYI     | Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay $\tau$ . See Remark 2. (Integer > 0 or blank)                          |
| DELAYR     | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)  |
| DPHASEI    | Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle $\theta$ . (See Remark 2. (Integer > 0 or blank)                        |
| DPHASER    | Value of phase angle $\theta$ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)                              |
| TC         | Set identification number of the TABLEDi entry that gives $C(f)$ . See Remark 2. (Integer > 0 or blank)   |
| RC         | Value of C to be used for all frequencies. See Remark 2.. (Real or blank)   |
| TD         | Set identification number of the TABLEDi entry that gives $D(f)$ . See Remark 2. (Integer > 0 or blank)   |
| RD         | Value of D to be used for all frequencies. See Remark 2.. (Real or blank)   |
| TYPE       | Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Default = 0)  |

**Remarks:**

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.
2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, TC/RC, or TD/RD fields are blank or zero, the corresponding  $\tau$ ,  $\theta$ ,  $C(f)$  or  $D(f)$  will be zero. Either TC/RC or TD/RD may be blank or zero, but not both.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

| TYPE                  | TYPE of Dynamic Excitation                |
|-----------------------|---|
| 0, L, LO, LOA or LOAD | Applied load (force or moment) (Default)  |
| 1, D, DI, DIS or DISP | Enforced displacement using SPC/SPCD data |
| 2, V, VE, VEL or VELO | Enforced velocity using SPC/SPCD data     |
| 3, A, AC, ACC or ACCE | Enforced acceleration SPC/SPCD data       |

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RLOAD2**

## Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form.

$$\{P(f)\} = \{A\} \cdot B(f) e^{i\{\phi(f) + \theta - 2\pi f \tau\}}$$

for use in frequency response problems.

**Format:**

| 1      | 2   | 3        | 4                 | 5                   | 6     | 7     | 8    | 9 | 10 |
|--------|-----|----------|-------------------|---------------------|-------|-------|------|---|----|
| RLOAD2 | SID | EXCITEID | DELAYI/<br>DELAYR | DPHASEI/<br>DPHASER | TB/RB | TP/RP | TYPE |   |    |

**Example:**

|        |   |   |    |     |   |  |  |  |  |
|--------|---|---|----|-----|---|--|--|--|--|
| RLOAD2 | 5 | 3 | 15 | 5.0 | 7 |  |  |  |  |
|--------|---|---|----|-----|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. See Remarks 1. and 3. (Integer > 0)  |
| EXCITEID   | Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$ . See Remarks 4. and 5. (Integer > 0) |
| DELAYI     | Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay $\tau$ . See Remark 2. (Integer > 0 or blank)                          |
| DELAYR     | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)  |
| DPHASEI    | Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle $\theta$ . (See Remark 2. (Integer > 0 or blank))                       |
| DPHASER    | Value of phase angle $\theta$ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)                              |
| TB         | Set identification number of the TABLEDi entry that gives $B(f)$ . (Integer > 0)  |
| RB         | Value of B to be used for all frequencies. (Real, non-zero)   |
| TP         | Set identification number of the TABLEDi entry that gives $\phi(f)$ in degrees. (Integer $\geq 0$ )   |
| RP         | Value of $\phi$ to be used for all frequencies. See Remark 2. (Real or blank)   |
| TYPE       | Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Defaults = 0)   |

**Remarks:**

- Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, or TP/RP fields are blank or zero, the corresponding  $\tau$ ,  $\theta$ , or  $\phi(f)$  will be zero.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

| TYPE                  | TYPE of Dynamic Excitation                |
|-----------------------|---|
| 0, L, LO, LOA or LOAD | Applied load (force or moment) (Default)  |
| 1, D, DI, DIS or DISP | Enforced displacement using SPC/SPCD data |
| 2, V, VE, VEL or VELO | Enforced velocity using SPC/SPCD data     |
| 3, A, AC, ACC or ACCE | Enforced acceleration SPC/SPCD data       |

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:
  - Excitation specified by TYPE is applied load
    - There is no LOADSET request in Case Control
 

EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.
    - There is a LOADSET request in Case Control
 

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.
  - Excitation specified by TYPE is enforced motion
    - There is no LOADSET request in Case Control
 

EXCITEID will reference SPCD entries.
    - There is a LOADSET request in Case Control
 

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**ROTHYBD**

Hybrid damping for rotors

Specifies hybrid damping data for rotors.

**Format:**

| 1       | 2        | 3        | 4        | 5        | 6        | 7        | 8        | 9        | 10 |
|---------|----------|----------|----------|----------|----------|----------|----------|----------|----|
| ROTHYBD | ROTORID1 | HYBDAMP1 | ROTORID2 | HYBDAMP2 | ROTORID3 | HYBDAMP3 | ROTORID4 | HYBDAMP4 |    |
|         | ROTORID5 | HYBDAMP5 | -etc.-   |          |          |          |          |          |    |

**Examples:**

|         |    |     |    |     |    |     |  |  |  |
|---------|----|-----|----|-----|----|-----|--|--|--|
| ROTHYBD | 1  | 15  |    |     |    |     |  |  |  |
| ROTHYBD | 10 | 100 | 20 | 200 | 30 | 300 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ROTORIDI   | Identification number of rotor. (Integer > 0). See Remarks 1 and 2.  |
| HYBDAMPi   | Identification number of a HYBDAMP entry defining hybrid modal damping data. (Integer > 0). See Remarks 1 and 2. |

**Remarks:**

1. ROTORIDI - HYBDAMPi pair values referencing non-existent rotors are ignored.
2. If there is no HYBDAMP entry defined in the data for a HYBDAMPi specified for a valid ROTORIDI, the program terminates the execution with an appropriate fatal error.
3. Hybrid damping can result in very densely populated damping matrix causing significant performance penalty.

**ROTOR****Rotor Model Definition**

Specifies list of grids, elements or properties that comprise the rotor 3D model.

**Format:**

| 1     | 2       | 3     | 4    | 5    | 6    | 7 | 8 | 9 | 10 |
|-------|---------|-------|------|------|------|---|---|---|----|
| ROTOR | ROTORID | FRAME |      |      |      |   |   |   |    |
|       | LTYPE   | ID1   | ID2  | ID3  | etc. |   |   |   |    |
|       | AXIS    | GID1  | GID2 | etc. |      |   |   |   |    |

**Example:**

|       |      |     |      |    |  |  |  |  |  |
|-------|------|-----|------|----|--|--|--|--|--|
| ROTOR | 10   | ROT |      |    |  |  |  |  |  |
|       | ELEM | 10  | THRU | 12 |  |  |  |  |  |
|       | PROP | 1   | THRU | 5  |  |  |  |  |  |
|       | AXIS | 101 | 102  |    |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ROTORID    | Identification number of rotor. (Integer > 0).  |
| FRAME      | Analysis frame (Char, ROT or FIX, Required, Default: FIX)   |
| LTYPE      | ELEM or PROP or both, indicating whether the specified list references element IDs or property IDs. (Character; Required; No default) |
| Idi        | Note that the order is important. In case both ELEM and PROP are specified, ELEM should be specified first.                           |
| AXIS       | IDs of elements or properties comprising the rotor. (Integer > 0; Required; No default)   |
| GIDI       | Defines grid points which define the axis of rotation.  |
|            | IDs of grids comprising the axis of the rotor (Integer > 0; Required; No default)   |

**Remarks:**

- Supported element types for analysis in rotating reference frame:
  - 0D elements : CONM1, CONM2
  - 1D elements : CBEAM, CBAR
  - 2D elements : CQUAD4, CQUAD8, CTRIA3, CTRIA6
  - 3D elements : CHEXA, CPENTA, CTETRA, CPYRAM
- Supported element types for analysis in fixed reference frame:
  - 0D elements : CONM1, CONM2
  - 1D elements : CBEAM, CBAR

2D elements : CQUAD4, CQUAD8

3D elements : CHEXA, CPENTA, CTETRA, CPYRAM

3. Current limitation for analysis in fixed reference frame:
  - a. If shell elements are defined perpendicular to the rotor axis for a 3D rotor defined using ROTOR entry, then gyroscopic matrix is not available for those elements. For example, a flat disk like CD spinning about the center. If the shell elements are replaced by SOLID elements, then correct gyroscopic effects can be obtained.
  - b. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.
4. THRU option is supported in ROTOR entry. Note that the order is important for LTYPE. In case both ELEM and PROP are specified, ELEM should be specified first.
5. Analysis can be performed using coupled mass or diagonal mass for all the elements.
6. ROTORAX, ROTORG and ROTORSE should not be used along with ROTOR in ROT frame. They can only be used with ROTOR in FIX frame. (since ROTORAX, ROTORG and ROTORSE assume fixed reference frame.)
7. At least 2 grid points need to be defined on AXIS to complete ROTOR definition, these points may not be part of rotor.
8. For unbalance loads the grid point, at which UNBALNC is defined, should be part of ROTOR AXIS list.
9. In order to include CONM1/2 elements as part of a rotor, its element ID should be listed using ELEM in ROTOR definition.
10. Stator portion of the model should only be defined in residual for external superelement runs in rotating system.
11. Only permanent glue option is supported for contact analysis for rotating portion of the model.

**ROTORAX**

## Axisymmetric Model Rotor Definition

Specifies list of elements, properties or grid points that comprise an axisymmetric model rotor.

**Format:**

| 1       | 2       | 3     | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|---------|---------|-------|--------|-----|-----|-----|-----|-----|----|
| ROTORAX | ROTORID | LTYPE | ID1    | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|         | ID7     | ID8   | -etc.- |     |     |     |     |     |    |

or

|         |         |       |     |      |     |    |     |  |  |
|---------|---------|-------|-----|------|-----|----|-----|--|--|
| ROTORAX | ROTORID | LTYPE | ID1 | THRU | ID2 | BY | INC |  |  |
|---------|---------|-------|-----|------|-----|----|-----|--|--|

**Example:**

|         |     |      |    |      |    |  |  |  |  |
|---------|-----|------|----|------|----|--|--|--|--|
| ROTORAX | 100 | ELEM | 5  | THRU | 25 |  |  |  |  |
| ROTORAX | 10  | PROP | 10 |      |    |  |  |  |  |
| ROTORAX | 15  | GRID | 50 | 69   | 70 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ROTORID    | Identification number of rotor. (Integer > 0). See Remarks 1. and 2..   |
| LTYPE      | ELEM, PROP or GRID, indicating whether the specified list references element IDs, property IDs or grid point IDs, respectively. (Character; Required; No default). See Remarks 3. through 6.. |
| IDi        | IDs of elements, properties or grid points comprising the rotor. (Integer > 0; Required; No default)  |
| THRU       | Implies a range of identification numbers. (Optional).  |
| BY         | Implies that an increment is being specified for the THRU option (Optional)   |
| INC        | ID increment. (Integer > 0; Optional)   |

**Remarks:**

1. ROTORIDs of ROTORAX entries may not be the same as the ROTORIDs of ROTORG entries or ROTORSE entries.
2. Multiple ROTORAX entries with the same ROTORID are supported.
3. When the GRID option is used for the LTYPE field, a minimum of two grid points must be specified in the list. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. For every rotor defined by a ROTORAX entry, there should be at least one entry using the GRID option for the LTYPE field to define the axis of symmetry.

5. Grid points specified on ROTORAX entries using the GRID option for the LTYPE field must be collinear. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
6. Only grid points specified on ROTORAX entries using the GRID option for the LTYPE field may be referenced by RSPINR, RSPINT and UNBALNC entries.

**ROTORG**

## Rotor Line Model Grids

Specifies grids that compose the rotor line model.

**Format:**

| 1      | 2       | 3     | 4     | 5      | 6     | 7     | 8     | 9     | 10 |
|--------|---------|-------|-------|--------|-------|-------|-------|-------|----|
| ROTORG | ROTORID | GRID1 | GRID2 | GRID3  | GRID4 | GRID5 | GRID6 | GRID7 |    |
|        |         | GRID8 | GRID9 | -etc.- |       |       |       |       |    |

or

|        |         |       |      |       |    |     |  |  |  |
|--------|---------|-------|------|-------|----|-----|--|--|--|
| ROTORG | ROTORID | GRID1 | THRU | GRID2 | BY | INC |  |  |  |
|--------|---------|-------|------|-------|----|-----|--|--|--|

**Example:**

|        |     |     |      |     |      |  |  |  |  |
|--------|-----|-----|------|-----|------|--|--|--|--|
| ROTORG | 100 | 101 | 1002 | 103 | 4001 |  |  |  |  |
|--------|-----|-----|------|-----|------|--|--|--|--|

|        |     |      |      |      |    |   |  |  |  |
|--------|-----|------|------|------|----|---|--|--|--|
| ROTORG | 200 | 1001 | THRU | 1100 | BY | 2 |  |  |  |
|--------|-----|------|------|------|----|---|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ROTORID    | Identification number of rotor. (Integer > 0; Required). See Remarks 1. and 2..             |
| GRIDi      | Grids comprising the rotor. (Integer > 0; Required; no Default). See Remarks 3. through 7.. |
| THRU       | Specifies a range of identification numbers. (Optional)                                     |
| BY         | Specifies an increment for a THRU specification (Optional)                                  |
| INC        | Grid point number increment. (Integer)  |

**Remarks:**

1. ROTORIDs of ROTORG entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORSE entries.
2. Multiple ROTORG entries with the same ROTORID are supported.
3. Grid IDs must be unique. Duplicate grid IDs will produce a fatal error.
4. All grids specified on ROTORG entries for a specific ROTORID must be collinear. Collinearity will be checked.
5. If not using superelements, no element stiffness can be connected between any GRID listed on a ROTORG entry and any GRID not listed on the ROTORG. In this case, any connections to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.
6. If superelements are used, no element stiffness in the residual structure may connect between a GRID listed on a ROTORG and any GRID not listed on the ROTORG. In this case, any connections in the residual structure to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.

7. All mass for any rotor defined using a ROTORG entry should be defined on the GRIDs listed on the ROTORG entry. Any mass which should belong to the rotor, but is placed on GRIDs not listed on the ROTORG entry, will be ignored when calculating the gyroscopic terms for the rotor. Only the mass shown in the ROTOR DYNAMICS MASS SUMMARY in the .f06 file is included in the calculation of the gyroscopic terms for the rotor.
8. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.

**ROTORSE**

## Rotor Superelement Identification

Specifies grids that compose the rotor line model. An alternate to the ROTORG entry when superelements are used.

**Format:**

| 1       | 2       | 3    | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|---------|------|-------|---|---|---|---|---|----|
| ROTORSE | ROTORID | SEID | SEOPT |   |   |   |   |   |    |

**Example:**

|         |    |   |  |  |  |  |  |  |  |
|---------|----|---|--|--|--|--|--|--|--|
| ROTORSE | 10 | 1 |  |  |  |  |  |  |  |
|---------|----|---|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ROTORID    | Identification number of rotor line model. (Integer > 0; Required). See Remark 1.                       |
| SEID       | Superelement identification number of rotor superelement. (Integer > 0; Required)                       |
| SEOPT      | Form of superelement for calculation of gyroscopic terms. (Integer = 1 or 2; Default = 1) See Remark 3. |

**Remarks:**

1. ROTORIDs of ROTORSE entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORG entries.
2. A ROTORSE entry can be used when the rotor is placed in a superelement.
3. The rotor line model may be the boundary of a 3D rotor superelement or the rotor line model may be a superelement itself. SEOPT is chosen to distinguish between these cases. The options are:
  - 1- If the user has a 3D model of the rotor and places it in a superelement with ID SEID, the boundary (a-set) of this superelement must consist of no more and no less than the collinear rotor line model. This will be checked. Specify SEOPT as 1 to identify this configuration. When this option is used, the A-set matrices of the superelement are used to calculate the gyroscopic terms. This option should be used for external superelements.
  - 2- If the user has a line model of the rotor and places it in a superelement with ID SEID, this superelement (g-set) may be partially or completely reduced in the SE reduction process. This superelement must consist of no more and no less than the rotor line model. Specify SEOPT as 2 to identify this configuration. When this option is used, the G-set matrices of the superelement are used to calculate the gyroscopic terms.
4. Rotors specified using the ROTORSE entry can be connected directly to the support structure. In contrast, rotors specified using the ROTORG entry must employ rigid elements to keep the rotor disconnected from the support in the G-set of the residual structure.
5. Static and component mode reduction of the rotor line model are supported when using ROTORSE entries.



**RROD****Rigid Pin-Ended Element Connection**

Defines a pin-ended element that is rigid in translation.

**Format:**

| 1    | 2   | 3  | 4  | 5   | 6   | 7     | 8 | 9 | 10 |
|------|-----|----|----|-----|-----|-------|---|---|----|
| RROD | EID | GA | GB | CMA | CMB | ALPHA |   |   |    |

**Example:**

|      |    |   |   |   |  |       |  |  |  |
|------|----|---|---|---|--|-------|--|--|--|
| RROD | 14 | 1 | 2 | 2 |  | 6.5-6 |  |  |  |
|------|----|---|---|---|--|-------|--|--|--|

| Describer | Meaning  |
|-----------|--|
| EID       | Element identification number. (0 < Integer < 100,000,000)   |
| GA, GB    | Grid point identification numbers of connection points. (Integer > 0)  |
| CMA, CMB  | Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear RROD. For Lagrange RROD, both CMA and CMB can be blank.) See Remark 3. |
| ALPHA     | Thermal expansion coefficient. See Remark 11. (Real or blank)  |

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally one Lagrange multiplier degree-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points.
- For the Lagrange method, if both CMA and CMB are blanks, Nastran will compute the best degree-of-freedom for the dependent degree-of-freedom.
- The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- Element identification numbers should be unique with respect to all other element identification numbers.
- RROD, among other eligible rigid element types, can be selected via MPC and SET3.
- Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
- Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.

9. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the element. This implies that the element must have finite length.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RROD.

**RSPINR****Relative Spin Rates Between Rotors**

Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis and rotor damping parameters.

**Format:**

| 1      | 2       | 3       | 4       | 5      | 6     | 7        | 8 | 9 | 10 |
|--------|---------|---------|---------|--------|-------|----------|---|---|----|
| RSPINR | ROTORID | GRIDA   | GRIDB   | SPDUNT | SPTID | ROTRSEID |   |   |    |
|        | GR      | ALPHAR1 | ALPHAR2 | WR3R   | WR4R  | WRHR     |   |   |    |

**Example:**

| 1      | 2    | 3    | 4    | 5     | 6     | 7     | 8 | 9 | 10 |
|--------|------|------|------|-------|-------|-------|---|---|----|
| RSPINR | 100  | 1001 | 1002 | RPM   | 100   | 5     |   |   |    |
|        | 0.02 | 1    | 3    | 1000. | 1500. | 1200. |   |   |    |

| Descriptor  | Meaning  |
|-------------|--|
| ROTORID     | Identification number of rotor. (Integer > 0; Required). See Remark 1.   |
| GRIDA/GRIDB | Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4.<br>(Integer > 0; Required)   |
| SPDUNIT     | Specifies whether the listing of relative spin rates is given in terms of RPM<br>(revolutions/minute) or frequency (revolutions (cycles)/sec). (Character; 'RPM' or<br>'FREQ'; Required) |
| SPTID       | Table for relative rotor spin rates. See Remark 5. (Real or Integer, if integer, must be<br>> 0; Required)   |
| GR          | Rotor structural damping factor. See Remarks 6. and 8. (Real; Default = 0.0)   |
| ALPHAR1     | Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remarks 7.<br>and 8. (Real; Default = 0.0)   |
| ALPHAR2     | Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remarks<br>7. and 8. (Real; Default = 0.0)  |
| ROTRSEID    | Identification number of the superelement in which the rotor specified in the<br>ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.                                    |
| WR3R        | Specifies "average" excitation frequency for calculation of rotor damping and<br>circulation terms for rotor structural damping specified through GR field. (Real,<br>Default = 0.)      |

| Descriptor | Meaning  |
|------------|--|
| WR4R       | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)    |
| WRHR       | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.) |

**Remarks:**

1. The ROTORID-ROTRSEID pair must be unique across all RSPINR entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINR must be part of the SEs ASET during creation run. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.  
If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.
5. If SPTID is a real number, the value is considered constant. If SPTID is an integer number, the value references a DDVAL entry that specifies the relative rotor spin rates. The number of spin rates for each rotor must be the same. Relative spin rates are determined by correlation of table entries. The ith entry for each rotor specifies the relative spin rates between rotors at RPMi/FREQi. Spin rates for the reference rotor must be in ascending or descending order.
6. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping or structural damping depending on the solution. That is,

$$[B_{rotor}]_{structural} = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

where WR3 is a user parameter, or

$$[K_{rotor}] = (1 + iGR)[K_{rotor}]$$

depending on the solution sequence, SYNC/ASYNC and value of PARAM, GYROAVG. See Remark for all the damping and circulation terms added to the equation in the different cases.

In case WR3R is specified in RSPINR, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

7. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$

8. The various rotor damping and circulation matrices applicable to frequency response and complex eigenvalue solutions are shown in the following table.

| Solution  | Damping   | Circulation  |
|---|---|--|
| Frequency Response -<br>ASYNC option                        | $i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$  | $\Omega_R(\Omega_{ref}) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] + \left(\frac{1}{\omega}\right)[KH_R^C] \end{pmatrix}$ |
| Frequency Response -<br>ASYNC option w/<br>PARAM,GYROAVG,-1 | $i\omega \begin{pmatrix} [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{pmatrix}$ | $\Omega_R(\Omega_{ref}) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{pmatrix}$       |
| Frequency Response -<br>SYNC option                         | $i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$  | $\Omega_R(\omega) \begin{pmatrix} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] \\ + \left(\frac{1}{\omega}\right)[KH_R^C] \end{pmatrix}$    |

| Solution  | Damping   | Circulation  |
|---|---|--|
| Frequency Response - SYNC option w/<br>PARAM,GYROAVG,-1 | $i\omega \left( [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left( \frac{GR}{WR3} \right) [K_R] + \left( \frac{1}{WR4} \right) [K4_R] + \left( \frac{1}{WRH} \right) [KH_R]$ | $\omega \beta_R \left( [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left( \frac{GR}{WR3} \right) [K_R^C] + \left( \frac{1}{WR4} \right) [K4_R^C] + \left( \frac{1}{WRH} \right) [KH_R^C]$<br>$+ \alpha_R \left( [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left( \frac{GR}{WR3} \right) [K_R^C] + \left( \frac{1}{WR4} \right) [K4_R^C] + \left( \frac{1}{WRH} \right) [KH_R^C]$ |
| Complex Modes - ASYNC option                            | $i\omega \left( [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left( \frac{GR}{WR3} \right) [K_R] + \left( \frac{1}{WR4} \right) [K4_R] + \left( \frac{1}{WRH} \right) [KH_R]$ | $\Omega_R(\Omega_{ref}) \left( [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left( \frac{GR}{WR3} \right) [K_R^C] + \left( \frac{1}{WR4} \right) [K4_R^C] + \left( \frac{1}{WRH} \right) [KH_R^C]$   |
| Complex Modes - SYNC option                             | $i\omega \left( [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \right) + \left( \frac{GR}{WR3} \right) [K_R] + \left( \frac{1}{WR4} \right) [K4_R] + \left( \frac{1}{WRH} \right) [KH_R]$ | $\omega \beta_R \left( [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left( \frac{GR}{WR3} \right) [K_R^C] + \left( \frac{1}{WR4} \right) [K4_R^C] + \left( \frac{1}{WRH} \right) [KH_R^C]$<br>$+ \alpha_R \left( [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \right) + \left( \frac{GR}{WR3} \right) [K_R^C] + \left( \frac{1}{WR4} \right) [K4_R^C] + \left( \frac{1}{WRH} \right) [KH_R^C]$ |

where:

- $[B_R]$  = the rotor viscous damping
- $[M_R]$  = the rotor mass
- $[K_R]$  = the rotor stiffness
- $[K4_R]$  = the rotor material damping
- $[BH_R]$  = the rotor viscous hybrid damping
- $[KH_R]$  = the rotor structural hybrid damping
- $[B_R^C]$  = the circulation due to rotor viscous damping
- $M_R^C$  = the circulation due to rotor 'mass'
- $K_R^C$  = the circulation due to rotor structural 'stiffness'
- $[K4_R^C]$  = the circulation due to rotor material damping
- $[BH_R^C]$  = the circulation due to rotor viscous hybrid damping
- $[KH_R^C]$  = the circulation due to rotor structural hybrid damping
- $\alpha 1_R, \alpha 2_R$  = used to specify Rayleigh viscous damping  
 $([B_R]_{Rayleigh} = \alpha 1_R[M_R] + \alpha 2_R[K_R])$

- $\alpha_R, \beta_R$  = scale factors of linear fit of rotor speed to reference rotor speed. The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor.
- $WR3, WR4, WRH$  = User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.
- a. Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- b. For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.
- c. All rotor damping terms are cumulative. Multiple damping options should be selected with caution.

**RSPINT****Rotor Spin Rates**

Specifies rotor spin rates for transient analysis.

**Format:**

| 1      | 2       | 3       | 4       | 5      | 6     | 7      | 8       | 9 | 10 |
|--------|---------|---------|---------|--------|-------|--------|---------|---|----|
| RSPINT | ROTORID | GRIDA   | GRIDB   | SPDUNT | SPTID | SPDOUT | ROTSEID |   |    |
|        | GR      | ALPHAR1 | ALPHAR2 | WR3R   | WR4R  | WRHR   |         |   |    |

**Example:**

| 1      | 2    | 3    | 4     | 5     | 6     | 7     | 8 | 9 | 10 |
|--------|------|------|-------|-------|-------|-------|---|---|----|
| RSPINT | 100  | 1001 | 1002  | RPM   | 1001  |       |   |   |    |
|        | 0.01 | 0.01 | 0.002 | 1000. | 1500. | 1200. |   |   |    |

| Descriptor  | Meaning   |
|-------------|---|
| ROTORID     | Identification number of rotor. (Integer > 0; Required). See Remark 1.  |
| GRIDA/GRIDB | Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4. (Integer > 0; Required)   |
| SPDUNIT     | Specifies whether the spin rates are given in terms of RPM (revolutions/minute) or frequency (revolutions(cycles)/sec). (Character; 'RPM' or 'FREQ'; Required)                |
| SPTID       | Rotor spin rate. See Remark 5. (Integer > 0; Required)  |
| SPDOUT      | EPOINT to output the rotor speed vs. time. Output will be in SPDUNITs (Integer > 0 or blank)  |
| GR          | Rotor structural damping factor. See Remark 6. and 8. (Real; Default = 0.0)   |
| ALPHAR1     | Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)  |
| ALPHAR2     | Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)   |
| ROTRSEID    | Identification number of the superelement in which the rotor specified in the ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.                            |
| WR3R        | Specifies "average" excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field. (Real, Default = 0.) |

| Descriptor | Meaning  |
|------------|--|
| WR4R       | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)    |
| WRHR       | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.) |

**Remarks:**

1. The ROTORID-ROTRSEID pair must be unique across all RSPINT entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINT must be part of the SEs ASET during creation. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.  
If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.
5. SPTID references a TABLED1 entry that specifies the rotor spin rate history.
6. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}]_{structural} = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

where WR3 is a user parameter.

In case WR3R is specified in RSPINT, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

7. Rayleigh damping for the rotor will be calculated as
$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$
8. The various rotor damping and circulation matrices used in transient analysis are shown in the following table.

| Damping   | Circulation (added to stiffness)  |
|---|---|
| $[B_R] + [BH_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R]$ $+ \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R]$ $+ \left(\frac{1}{WRH}\right)[KH_R]$ | $\Omega_R(t) \begin{pmatrix} [B_R^C] + [BH_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{pmatrix}$ |

where:

|                          |  |
|--------------------------|--|
| $[B_R]$                  | = the rotor viscous damping  |
| $[M_R]$                  | = the rotor mass   |
| $[K_R]$                  | = the rotor stiffness  |
| $[K4_R]$                 | = the rotor material damping   |
| $[BH_R]$                 | = the rotor viscous hybrid damping   |
| $[KH_R]$                 | = the rotor structural hybrid damping  |
| $[B_R^C]$                | = the circulation due to rotor viscous damping   |
| $M_R^C$                  | = the circulation due to rotor 'mass'  |
| $K_R^C$                  | = the circulation due to rotor structural 'stiffness'  |
| $[K4_R^C]$               | = the circulation due to rotor material damping  |
| $[BH_R^C]$               | = the circulation due to rotor viscous hybrid damping  |
| $[KH_R^C]$               | = the circulation due to rotor structural hybrid damping   |
| $\alpha 1_R, \alpha 2_R$ | = used to specify Rayleigh viscous damping<br>$([B_R]_{Rayleigh} = \alpha 1_R[M_R] + \alpha 2_R[K_R])$   |
| $\alpha_R, \beta_R$      | = scale factors of linear fit of rotor speed to reference rotor speed.<br>The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor. |
| $WR3, WR4, WRH$          | = User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.   |

- a. Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- b. For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.

- c. All rotor damping terms are cumulative. Multiple damping options should be selected with caution.

**RSPLINE****Interpolation Constraint Element**

Defines multipoint constraints for the interpolation of displacements at grid points.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6      | 7  | 8  | 9  | 10 |
|---------|-----|-----|----|----|--------|----|----|----|----|
| RSPLINE | EID | D/L | G1 | G2 | C2     | G3 | C3 | G4 |    |
|         | C4  | G5  | C5 | G6 | -etc.- |    |    |    |    |

**Example:**

|         |     |     |     |    |        |    |  |    |  |
|---------|-----|-----|-----|----|--------|----|--|----|--|
| RSPLINE | 73  | .05 | 27  | 28 | 123456 | 29 |  | 30 |  |
|         | 123 | 75  | 123 | 71 |        |    |  |    |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| D/L        | Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default = 0.1) |
| Gi         | Grid point identification number. (Integer > 0)  |
| Ci         | Components to be constrained. See Remark 2. (Blank or any combination of the Integers 1 through 6.)              |

**Remarks:**

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points. This is a linear method only element, and not controlled with the Case Control command RIGID.
2. A blank field for Ci indicates that all six degrees-of-freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a Gi, not a Ci. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.
3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.
5. EIDs must be unique.
6. RSPLINE, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
11. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results.
12. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.
13. The RSPLINE has a limit of 100 grid points.

**RSSCON**

## Shell-to-Solid Element Connector

Defines multipoint constraints to model clamped connections of shell-to-solid elements.

**Format:**

| 1      | 2    | 3    | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|------|------|-----|-----|-----|-----|-----|-----|----|
| RSSCON | RBID | TYPE | ES1 | EA1 | EB1 | ES2 | EA2 | EB2 |    |

**Examples:**

|        |     |      |     |     |    |    |    |    |  |
|--------|-----|------|-----|-----|----|----|----|----|--|
| RSSCON | 110 | GRID | 11  | 12  | 13 | 14 | 15 | 16 |  |
| RSSCON | 111 | GRID | 31  | 74  | 75 |    |    |    |  |
| RSSCON | 115 | ELEM | 311 | 741 |    |    |    |    |  |

| Descriptor | Meaning   |
|------------|---|
| RBID       | Element identification number. (0 < Integer < 100,000,000)  |
| TYPE       | Type of connectivity:<br>ELEM      Connection is described with element identification numbers.<br>GRID      Connection is described with grid point identification numbers.<br>(Character: “GRID” or “ELEM”; Default = “ELEM”) |
| ES1        | Shell element identification number if TYPE = “ELEM”. Shell grid point identification number if TYPE = “GRID”. See <a href="#">Figure 9-153</a> . (Integer > 0)   |
| EA1        | Solid element identification number if TYPE = “ELEM”. Solid grid point identification number if TYPE = “GRID”. (Integer > 0)  |
| EB1        | Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)   |
| ES2        | Shell grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)   |
| EA2        | Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)   |
| EB2        | Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)   |

**Remarks:**

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of-freedom are put in the dependent set (m-set). The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower solid edge. The two rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson’s ratio effects are considered in the translational degrees-of-freedom.

2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see PARAM,TOLRSC and PARAM,SEPIXOVR.
3. When using the TYPE = "ELEM" option
  - The elements may be p-elements or h-elements. The solid elements are CHEXA, CPENTA, and CTETRA with and without midside nodes. The shell elements are CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8, or CTRIA6. If the solid element has mid-side nodes, then the shell element needs mid-side nodes on the common edge.
  - In case of p-elements, the p-value of the shell element edge is adjusted to the higher of the p-value of the upper or lower solid p-element edge. If one of the elements is an h-element, then the p-value of the adjacent edge is lowered to 1.
  - Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using SEELT entry to reassign the downstream boundary element to an upstream superelement.
  - When a straight shell p-element edge and a solid p-element are connected, the geometry of the shell edge is not changed to fit the solid face. When a curved shell p-element edge and a solid p-element are connected, the two solid edges and solid face are not changed to match the shell edge.
  - It is not recommended to connect more than one shell element to the same solid using the ELEM option. If attempted, conflicts in the multipoint constraint relations may lead to UFM 6692.
4. When using TYPE = "GRID" option
  - The GRID option does not verify that the grids used are valid shell and/or solid grids.
  - The hierarchical degrees-of-freedom of p-element edges are not constrained. The GRID option is therefore not recommended for p-elements.
  - The grids in the GRID option can be in different superelements. The shell grid must be in the upstream superelement.
5. It is recommended that the height of the solid element's face is approximately equal to the shell element's thickness of the shell. The shell edge should then be placed in the middle of the solid face.
6. The shell edge may coincide with the upper or lower edge of the solid face.
7. RSSCON, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
9. The RSSCON is ignored in heat-transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

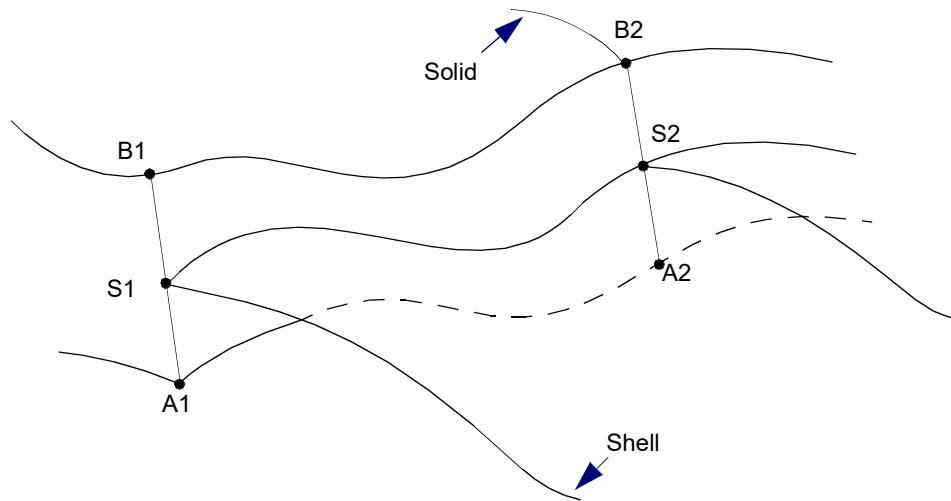


Figure 9-153 Shell Elements Connected to the Faces of Solid Elements

**RTRPLT****Rigid Triangular Plate**

Defines a rigid triangular plate.

**Format:**

| 1      | 2   | 3   | 4   | 5     | 6   | 7   | 8   | 9 | 10 |
|--------|-----|-----|-----|-------|-----|-----|-----|---|----|
| RTRPLT | EID | GA  | GB  | GC    | CNA | CNB | CNC |   |    |
|        | CMA | CMB | CMC | ALPHA |     |     |     |   |    |

**Example:**

|        |   |   |   |   |      |   |   |  |  |
|--------|---|---|---|---|------|---|---|--|--|
| RTRPLT | 7 | 1 | 2 | 3 | 1236 | 3 | 3 |  |  |
|--------|---|---|---|---|------|---|---|--|--|

| Descriptor    | Meaning   |
|---------------|---|
| EID           | Element identification number. (0 < Integer < 100,000,000)  |
| GA, GB, GC    | Grid point identification number of connection points.  |
| CNA, CNB, CNC | Independent degrees-of-freedom in the global coordinate system for the element at grid points GA, GB, and GC, indicated by any of the Integers 1 through 6 with no embedded blanks. See Remark 3. (Integer $\geq 0$ or blank) |
| CMA, CMB, CMC | Component numbers of dependent degrees-of-freedom in the global coordinate system. (Any of the Integers 1 through 6 with no embedded blanks, or 0 or blank.)  |
| ALPHA         | Thermal expansion coefficient. See Remark 12. (Real or blank)   |

**Remarks:**

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
3. For the linear method, the total number of components in CNA, CNB, and CNC must equal six; for example, CNA = 1236, CNB = 3, CNC = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA= 123456 or CNB = 123456 or CNC = 123456 is allowed. For this type of element, RTRPLT1 gives a simpler input format.
4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.

5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RTRPLT, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) (p. 155) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
12. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is defined as the following, the bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.  
  
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RTRPLT.

**RTRPLT1****Rigid Triangular Plate (Alternative Format)**

Alternative format to define a rigid triangular plate element connecting three grid points.

**Format:**

| 1       | 2   | 3  | 4  | 5  | 6   | 7   | 8     | 9 | 10 |
|---------|-----|----|----|----|-----|-----|-------|---|----|
| RTRPLT1 | EID | GA | GB | GC | CMB | CMC | ALPHA |   |    |

**Example:**

|         |   |   |   |   |      |   |       |  |  |
|---------|---|---|---|---|------|---|-------|--|--|
| RTRPLT1 | 7 | 1 | 2 | 3 | 1236 | 3 | 6.0-6 |  |  |
|---------|---|---|---|---|------|---|-------|--|--|

| Descriptor | Meaning  |
|------------|--|
| EID        | Element identification number. (0 < Integer < 100,000,000)   |
| GA, GB, GC | Grid point identification number of connection points. (Integer > 0)   |
| CMB, CMC   | Component numbers at GB and GC in the global coordinate systems, which are constrained to move with the rigid body. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.) |
| ALPHA      | Thermal expansion coefficient. See Remark 9. (Real or blank)   |

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- RTRPLT1 is a preferred input format for the Lagrange method.
- When CMB = “123456” or blank, CMC = “123456” or blank, the grid points GB and BC are constrained to move with GA as a rigid triangular plate. For default, CMB = “123456” and CMC = “123456”. Any number of degrees-of-freedom at grid points GB and GC can be released not to move with the rigid body.
- The length of any two connected grid points must be greater than zero.
- For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA, GB, and GC will be computed even if these three points have different global coordinate systems.

7. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.  
There is no current entry for a TREF, so if TEMP(INIT) is not specified in Case Control, a reference temperature of 0.0 is used for the RTRPLT1.
8. Element identification numbers should be unique with respect to all other element identification numbers.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.

**RVDOF**

## Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

**Format:**

| 1     | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|-------|-----|----|-----|----|-----|----|-----|----|----|
| RVDOF | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|       |     |   |     |   |  |  |  |  |  |
|-------|-----|---|-----|---|--|--|--|--|--|
| RVDOF | 800 | 1 | 850 | 2 |  |  |  |  |  |
|-------|-----|---|-----|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar identification number. (Integer > 0)  |
| Ci         | Component numbers. (Any one of the integers 1 through 6 for grid points and integer zero or blank for scalar points) |

**Remarks:**

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

**RVDOF1**

## Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

**Format:**

| 1      | 2   | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|--------|-----|-----|-----|-----|-----|----|
| RVDOF1 | C   | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|        | ID8 | ID9 | -etc.- |     |     |     |     |     |    |

**Example:**

|        |   |     |     |     |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|
| RVDOF1 | 3 | 100 | 210 | 450 |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|

**Alternate Format and Example:**

|        |    |     |        |     |  |  |  |  |  |
|--------|----|-----|--------|-----|--|--|--|--|--|
| RVDOF1 | C  | ID1 | "THRU" | ID2 |  |  |  |  |  |
| RVDOF1 | 12 | 6   | THRU   | 21  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.) |
| IDi        | Grid or scalar identification numbers. (Integer > 0 or "THRU"; for "THRU" option, ID1 < ID2)   |

**Remarks:**

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.
3. If the alternate format is used, points in the sequence ID1 through ID2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
4. When alternative format is used, single point IDs cannot be specified.

## SANGLE

Analytical contact threshold angle in SOL 600

Defines automatic analytical contact threshold angle for multiple subcases in SOL 600 only.

### Format:

| 1      | 2   | 3   | 4     | 5   | 6   | 7     | 8 | 9 | 10 |
|--------|-----|-----|-------|-----|-----|-------|---|---|----|
| SANGLE | IDC | IDB | Angle | IDC | IDB | Angle |   |   |    |

### Example:

|        |   |   |      |   |   |      |  |  |  |
|--------|---|---|------|---|---|------|--|--|--|
| SANGLE | 1 | 4 | 50.0 | 1 | 6 |      |  |  |  |
|        | 2 | 4 | -1.0 | 2 | 6 | 55.0 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| IDC        | Identification number of a SUBCASE Case Control command. (Integer; no Default) To enter a value corresponding to Marc's increment zero, set IDC=0. |
| IDB        | Identification of a contact body (must be the same as a BCBODY ID) (Integer; no Default)   |
| Angle      | Threshold automatic analytical contact angle (SANGLE). (Real; Default = 60.0)<br>A value of -1.0 turns off analytical.                             |

### Remarks:

1. This entry should only be made if IDSPL=1 and if SANGLE is a non-zero integer value on one or more BCBODY entry.
2. This entry is available in SOL 600 only.
3. For the example, BCBODY with id=4 has a threshold angle of 50.0 degrees in subcase 1 and analytical contact is turned off in subcase 2. For bcbbody=6, the analytical contact is on for subcaes 1 and 2 and the threshold angle is 60.0 degrees (the default) and 55.0 degrees for subcases 1 and 2 respectively.
4. Only those contact bodies whose SANGLE changes from subcase to subcase or is turned on/off need be described here. Those with constant SANGLE may be described on the BCBODY entry.

**SEBNDRY**

## Superelement Boundary-Point Definition

Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.

**Format:**

| 1       | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| SEBNDRY | SEIDA | SEIDB | GIDA1  | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | -etc.- |       |       |       |       |       |    |

**Example 1:**

|         |     |   |    |    |    |    |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|
| SEBNDRY | 400 | 4 | 10 | 20 | 30 | 40 |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|

**Example 2:**

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| SEBNDRY | 400 | ALL | 10 | 20 | 30 | THRU | 40 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEIDA      | Partitioned superelement identification number. (Integer > 0)  |
| SEIDB      | Superelement identification. See Remark 2. (Integer > 0 or Character “ALL”; Default = “ALL”)                               |
| GIDAi      | Identification number of a boundary grid point in superelement SEIDA. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.) |

**Remarks:**

1. SEBNDRY may only be specified in the main Bulk Data Section and is not recognized after the BEGIN SUPER=n.
2. SEIDB may reference partitioned superelements or superelements in the main Bulk Data Section.

**SEBSET**

## Fixed Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|--------|------|-----|----|-----|----|-----|----|---|----|
| SEBSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|        |   |   |     |    |   |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|
| SEBSET | 5 | 2 | 135 | 14 | 6 |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. (Integer > 0)   |
| Ci         | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points) |
| IDi        | Grid or scalar point identification numbers. (Integer > 0)  |

**Remarks:**

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If only SEBSETi are entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If both SEBSETi and SECSETi entries are present, the c-set degrees-of-freedom are defined by the SECSETi entries and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SEBSET1**

## Fixed Boundary Degree-of-Freedom, Alternate Form of SEBSET

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode calculations.

**Format:**

| 1       | 2    | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|----|--------|----|----|----|----|----|
| SEBSET1 | SEID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|         | G7   | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|         |     |     |     |    |   |    |    |    |  |
|---------|-----|-----|-----|----|---|----|----|----|--|
| SEBSET1 | 5   | 2   | 135 | 14 | 6 | 23 | 24 | 25 |  |
|         | 122 | 127 |     |    |   |    |    |    |  |

**Alternate Format and Example:**

|         |      |   |    |        |    |  |  |  |  |
|---------|------|---|----|--------|----|--|--|--|--|
| SEBSET1 | SEID | C | G1 | "THRU" | G2 |  |  |  |  |
| SEBSET1 | 5    | 3 | 6  | THRU   | 32 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. (Integer > 0)   |
| C          | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points, 0 or blank for scalar points.) |
| Gi         | Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for THRU option G1 < G2.)  |

**Remarks:**

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If there are only SEBSETi entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both SEBSETi and SECSETi entries present, the c-set degrees-of-freedom are defined by the SECSETi entries, and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SEBULK**

## Partition Superelement Connection

Defines superelement boundary search options and a repeated, mirrored, or collector superelement.

**Format:**

| 1      | 2    | 3    | 4     | 5      | 6   | 7   | 8      | 9 | 10 |
|--------|------|------|-------|--------|-----|-----|--------|---|----|
| SEBULK | SEID | TYPE | RSEID | METHOD | TOL | LOC | UNITNO |   |    |

**Example:**

|        |    |        |   |      |        |  |  |  |  |
|--------|----|--------|---|------|--------|--|--|--|--|
| SEBULK | 14 | REPEAT | 4 | AUTO | 1.0E-3 |  |  |  |  |
|--------|----|--------|---|------|--------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned superelement identification number. (Integer > 0). See Remark 11.  |
| TYPE       | Superelement type. (Character; no Default)   |
|            | PRIMARY Primary. See Remarks 2. and 5.   |
|            | COLLCTR Collector. See Remark 3.   |
|            | EXTERNAL External. See Remarks 4. and 5.   |
|            | EXTOP2 External using an OUTPUT2 file created in an earlier run. See Remarks 4. and 5.   |
|            | EXTOP4 External using an OUTPUT4 file created in an earlier run. See Remarks 4. and 5.   |
|            | REPEAT Identical. See Remark 6.  |
|            | MIRROR Mirror. See Remark 6.   |
| RSEID      | Identification number of the reference superelement, used only if TYPE = "REPEAT" or "MIRROR". (Integer $\geq 0$ ; Default = 0). See Remark 6. |
| METHOD     | Method to be used when searching for boundary grid points. (Character: "AUTO" or "MANUAL"; Default = "AUTO"). See Remarks 7. and 8.            |
| TOL        | Location tolerance to be used when searching for boundary grid points. (Real; Default = 10E-5). See Remarks 10. and 11.                        |
| LOC        | Coincident location check option for manual connection option. (Character: "YES" or "NO"; Default = "YES"). See Remark 10.                     |
| UNITNO     | FORTRAN unit number for the OUTPUT2 or OUTPUT4 file (applicable and meaningful only when TYPE = "EXTOP2" or "EXTOP4"). See Remarks 4. and 5.   |

**Remarks:**

1. This specification must be specified in the MAIN Bulk Data and is meaningful only if part superelements (using BEGIN SUPER) or external superelements created by employing the EXTSEOUT Case Control command exist in the data.

2. TYPE = PRIMARY indicates that the superelement is a part superelement (using BEGIN SUPER).
  3. TYPE = "COLLCTR" indicates a collector superelement which does not contain any grids or scalar points.
  4. TYPE = "EXTERNAL", "EXTOP2" or "EXTOP4" indicates an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. See discussion under the description of the EXTSEOUT Case Control command. (For employing external superelements using the old three-step procedure, see discussion under the description of parameter EXTDROUT in Chapter 5.)
  5. A superelement whose TYPE is "PRIMARY", "EXTERNAL" or "EXTOP2" may itself be re-positioned by the use of an SELOC or an SEMPLN entry or both.
    - If it references an SELOC entry, then the primary superelement will be positioned at the location implied by the SELOC entry.
    - If it references an SEMPLN entry, then a mirror image of the primary superelement will be positioned using the plane defined by the SEMPLN entry.
    - If it references both an SELOC entry and an SEMPLN entry, then a mirror image of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.
  6. TYPE = "REPEAT" and TYPE = "MIRROR" have many common features, but they also have some important differences. These are described below.
- The following comments apply to both TYPE= "REPEAT" and TYPE = "MIRROR":
- a. An SEID whose TYPE is "REPEAT" or "MIRROR" is referred to as a secondary superelement. The RSEID specified in this case is regarded as the primary superelement.
  - b. The primary superelement must be either a part superelement or an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. It must be defined via its own SEBULK entry wherein TYPE must be either "PRIMARY", "EXTERNAL" or "EXTOP2".
  - c. The primary superelement does not include superelements upstream of the primary superelement.
  - d. The primary superelement may be re-positioned as indicated in Remark 5.
  - e. If the primary superelement specified by RSEID is a part superelement, then the secondary superelement is a "G-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement can be different from those of its primary superelement.
  - f. If the primary superelement specified by RSEID is an external superelement resulting from the use of the EXTSEOUT Case Control command in an earlier job, then the secondary superelement is an "A-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement are set and are the same as those of its primary superelement.

The following comments apply only to TYPE= "REPEAT":

- g. The secondary superelement in this case may reference an SELOC entry, an SEMPLN entry or both or none.

If it references an SELOC entry, then an identical copy of its primary superelement will be positioned at the location implied by the SELOC entry.

If it references an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

If it references neither an SELOC entry nor an SEMPLN entry, then the secondary superelement will merely be a duplicate of the primary superelement positioned at the same location as the primary superelement. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message.

The following comments apply only to TYPE= "MIRROR":

- h. The secondary superelement in this case must reference an SEMPLN entry. (Otherwise, the program will terminate the execution with an appropriate user fatal message.) In addition, the secondary superelement may also reference an SELOC entry.

If the secondary superelement references only an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

7. METHOD = "MANUAL" requires SECONCT entries. SEBNDRY and SEEXCLD, which reference SEID, will produce a fatal message.
8. SECONCT, SEBNDRY, and SEEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
9. For combined automatic and manual boundary search, the METHOD = "AUTO" should be specified and connections should be specified on a SECONCT entry.
10. TOL and LOC are the default values that can be modified between two superelements by providing the required tolerance on the SECONCT entry.
11. A SEID=0 is valid for the residual part only if it is desired to set a TOL value for the residual.

**SECONCT**

## Partitioned Superelement Boundary-Point Connection

Explicitly defines grid and scalar point connection procedures for a partitioned superelement.

**Format:**

| 1       | 2     | 3     | 4     | 5     | 6     | 7     | 8      | 9 | 10 |
|---------|-------|-------|-------|-------|-------|-------|--------|---|----|
| SECONCT | SEIDA | SEIDB | TOL   | LOC   |       |       |        |   |    |
|         | GIDA1 | GIDB1 | GIDA2 | GIDB2 | GIDA3 | GIDB3 | -etc.- |   |    |

**Example:**

|         |      |      |        |     |      |      |  |  |  |
|---------|------|------|--------|-----|------|------|--|--|--|
| SECONCT | 10   | 20   | 1.0E-4 | YES |      |      |  |  |  |
|         | 1001 | 4001 |        |     | 2222 | 4444 |  |  |  |

**Alternate Format and Example:**

|         |       |        |       |       |        |       |  |  |  |
|---------|-------|--------|-------|-------|--------|-------|--|--|--|
| SECONCT | SEIDA | SEIDB  | TOL   | LOC   |        |       |  |  |  |
|         | GIDA1 | 'THRU' | GIDA2 | GIDB1 | 'THRU' | GIDB2 |  |  |  |
| SECONCT | 10    | 20     |       |       |        |       |  |  |  |
|         | 101   | 'THRU' | 110   | 201   | 'THRU' | 210   |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SEIDA      | Identification number of superelement for connection to SEIDB. (Integer $\geq 0$ )                            |
| SEIDB      | Identification number of superelement for connection to SEIDA. (Integer $\geq 0$ )                            |
| TOL        | Location tolerance to be used when searching for or checking boundary grid points.<br>(Real; Default = 10E-5) |
| LOC        | Coincident location check option for manual connection. (Character; "YES" or "NO";<br>Default = "YES")        |
| GIDAi      | Identification number of a grid or scalar point in superelement SEIDA, which will be connected to GIDBi.      |
| GIDBi      | Identification number of a grid or scalar point in superelement SEIDB, which will be connected to GIDAi.      |

**Remarks:**

1. SECONCT can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER = n command.
2. TOL and LOC can be used to override the default values specified on the SEBULK entries.
3. The continuation entry is optional.
4. The (GIDAi, GIDBi) pair must both be grids or scalar points.

5. All six degrees-of-freedom of grid points will be defined as boundary degrees-of-freedom.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.
7. Blank fields are allowed after the first GIDA1-GIDB1 pair. Blank fields must also occur in pairs. This remark does not apply to the alternate format.
8. For Alternate Format 1, the thru ranges must be closed sets. That is, all IDs listed between 101 and 110 in the example must exist in the model.

**SECSET**

## Free Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|--------|------|-----|----|-----|----|-----|----|---|----|
| SECSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|        |   |     |   |   |    |   |    |  |  |
|--------|---|-----|---|---|----|---|----|--|--|
| SECSET | 3 | 124 | 1 | 5 | 23 | 6 | 15 |  |  |
|--------|---|-----|---|---|----|---|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. (Integer > 0)  |
| Ci         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| IDi        | Grid or scalar point identification number. (Integer > 0)  |

**Remarks:**

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Coordinates listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined by the program and listed in the SEMAP table output.
2. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
3. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SECSET1**

Free Boundary Degree-of-Freedom, Alternate Form of SECSET

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

| 1       | 2    | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|----|--------|----|----|----|----|----|
| SECSET1 | SEID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|         | G7   | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|         |     |     |     |    |   |    |    |    |  |
|---------|-----|-----|-----|----|---|----|----|----|--|
| SECSET1 | 5   | 2   | 135 | 14 | 6 | 23 | 24 | 25 |  |
|         | 122 | 127 |     |    |   |    |    |    |  |

**Alternate Formats and Example:**

|         |      |   |       |        |    |  |  |  |  |
|---------|------|---|-------|--------|----|--|--|--|--|
| SECSET1 | SEID | C | G1    | "THRU" | G2 |  |  |  |  |
| SECSET1 | 5    | 3 | 6     | THRU   | 32 |  |  |  |  |
| SECSET1 | SEID |   | "ALL" |        |    |  |  |  |  |
| SECSET1 | SEID |   | ALL   |        |    |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. (Integer > 0)  |
| C          | Component numbers of degree-of-freedoms. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi         | Grid or scalar point identification number. (Integer > 0)  |

**Remarks:**

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSET*i* or SESUP entries. Degrees-of-freedom listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined automatically and listed in the SEMAP table output.
2. If the alternate formats are used, the grid points Gi are not required to exist or to be exterior degrees-of-freedom and may be listed on SECSET1 entries. Points of this type will cause one warning message but will otherwise be ignored.
3. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
5. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SECTAX**

Conical Shell Sector

Defines a sector of a conical shell.

**Format:**

| 1      | 2  | 3   | 4 | 5    | 6    | 7 | 8 | 9 | 10 |
|--------|----|-----|---|------|------|---|---|---|----|
| SECTAX | ID | RID | R | PHI1 | PHI2 |   |   |   |    |

**Example:**

|        |   |   |     |      |      |  |  |  |  |
|--------|---|---|-----|------|------|--|--|--|--|
| SECTAX | 1 | 2 | 3.0 | 30.0 | 40.0 |  |  |  |  |
|--------|---|---|-----|------|------|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Sector identification number. (Unique Integer > 0)          |
| RID        | Ring identification number. See RINGAX entry. (Integer > 0) |
| R          | Effective radius. (Real)                                    |
| PHI1, PHI2 | Azimuthal limits of sector in degrees. (Real)               |

**Remarks:**

1. SECTAX is allowed only if an AXIC entry is also present.
2. SECTAX identification numbers must be unique with respect to all other POINTAX, RINGAX and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

**SEDLINK****Multiple Design Variable Linking Across PART SE Boundary**

Relates one design variable of a PART SE to one or more other design variables from other PART SEs.

**Format:**

| 1       | 2      | 3     | 4     | 5      | 6     | 7      | 8    | 9  | 10 |
|---------|--------|-------|-------|--------|-------|--------|------|----|----|
| SEDLINK | ID     | DSEID | DDVID | C0     | CMULT | ISEID1 | IDV1 | C1 |    |
|         | ISEID2 | IDV2  | C2    | ISEID3 | IDV3  | C3     |      |    |    |
|         | ISEID4 | IDV4  | C4    | -etc.- |       |        |      |    |    |

**Example:**

|         |    |    |      |     |      |     |   |      |  |
|---------|----|----|------|-----|------|-----|---|------|--|
| SEDLINK | 10 | 10 | 2    | 0.1 | 0.33 | 8   | 8 | -1.0 |  |
|         | 11 | 6  | -1.0 | 20  | 8    | 7.0 |   |      |  |
|         | 11 | 2  | 2.0  |     |      |     |   |      |  |

| Descriptor         | Meaning  |
|--------------------|--|
| ID                 | Unique identification number. (Integer > 0)                      |
| DSEID              | PART SE identification number for DDVID (Integer $\geq 0$ )      |
| DDVID              | Dependent design variable identification number. (Integer > 0)   |
| C0                 | Constant term. (Real; Default = 0.0)                             |
| CMULT              | Constant multiplier. (Real; Default = 1.0)                       |
| ISEID <sub>i</sub> | PART SE identification number for IDVi (Integer $\geq 0$ )       |
| IDVi               | Independent design variable identification number. (Integer > 0) |
| C <sub>i</sub>     | Coefficient i corresponding to IDVi. (Real)                      |

**Remarks:**

- SEDLINK is provided for inter-PART SE design variables linking and must be placed in main Bulk Data Section for consideration. SEDLINK will be ignored if it shows up under 'BEGIN SUPER=seid' where seid>0.
- SEDLINK defines the relationship

$$DDVID = C0 + CMULT \sum_i C_i ISEID_i \cdot IDVi_{ISEID_i}$$

- This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
- CMULT provides a simple means of scaling the  $C_i$ . For example if  $C_i = 1/7, 2/7, 4/7$ , etc. is desired, then CMULT = 1/7 and  $C_i = 1, 2, 4$ , etc., may be input.

5. An independent  $IDV_{i,ISEID_i}$  must not occur on the same SEDLINK entry more than once.
6. ID is for user reference only.
7. If a design variable of a PART SE is specified as dependent on a SEDLINK entry, then it cannot be specified as independent on another SEDLINK or DLINK entry.

**SEDRSP2**

## Design Sensitivity Equation Response Quantities for PART SE

Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.

**Format:**

| 1       | 2         | 3       | 4            | 5       | 6       | 7       | 8      | 9  | 10 |
|---------|-----------|---------|--------------|---------|---------|---------|--------|----|----|
| SEDRSP2 | ID        | LABEL   | EQID or FUNC | REGION  | METHOD  | C1      | C2     | C3 |    |
|         | "DESVAR"  | DVSEID1 | DVID1        | DVSEID2 | DVID2   | DVSEID3 | DVID3  |    |    |
|         |           | DVSEID4 | DVID4        | -etc.-  |         |         |        |    |    |
|         | "DTABLE"  | LBSEID1 | LABL1        | LBSEID2 | LABL2   | LBSEID3 | LABL3  |    |    |
|         |           | LBSEID4 | LABL4        | -etc.-  |         |         |        |    |    |
|         | "DRESP1"  | R1SEID1 | NR1          | R1SEID2 | NR2     | R1SEID3 | NR3    |    |    |
|         |           | R1SEID4 | NR4          | -etc.-  |         |         |        |    |    |
|         | "DNODE"   | NDSEID1 | G1           | CMP1    | NDSEID2 | G2      | CMP2   |    |    |
|         |           | NDSEID3 | G3           | CMP3    | -etc.-  |         |        |    |    |
|         | "DVPREL1" | P1SEID1 | DPIP1        | P1SEID2 | DPIP2   | P1SEID3 | DPIP3  |    |    |
|         |           | P1SEID4 | DPIP4        | -etc.-  |         |         |        |    |    |
|         | "DVCREL1" | C1SEID1 | DCIC1        | C1SEID2 | DCIC2   | C1SEID3 | DCIC3  |    |    |
|         |           | C1SEID4 | DCIC4        | -etc.-  |         |         |        |    |    |
|         | "DVMREL1" | M1SEID1 | DMIM1        | M1SEID2 | DMIM2   | M1SEID3 | DMIM3  |    |    |
|         |           | M1SEID4 | DMIM4        | -etc.-  |         |         |        |    |    |
|         | "DVPREL2" | P2SEID1 | PDI2P1       | P2SEID2 | DPI2P2  | P2SEID3 | DPI2P3 |    |    |
|         |           | P2SEID4 | DPI2P4       | -etc.-  |         |         |        |    |    |
|         | "DVCREL2" | C2SEID1 | DC12C1       | C2SEID2 | DC12C2  | C2SEID3 | DC12C3 |    |    |
|         |           | C2SEID4 | DC12C4       | -etc.-  |         |         |        |    |    |
|         | "DVMREL2" | M2SEID1 | DMI2M1       | M2SEID2 | DMI2M2  | M2SEID3 | DMI2M3 |    |    |
|         |           | M2SEID4 | DMI2M4       | -etc.-  |         |         |        |    |    |

**Example:**

|         |         |       |     |    |     |   |   |  |  |
|---------|---------|-------|-----|----|-----|---|---|--|--|
| SEDRSP2 | 1       | LBUCK | 5   | 3  |     |   |   |  |  |
|         | DESVAR  | 0     | 101 | 1  | 3   | 1 | 4 |  |  |
|         |         | 10    | 201 |    |     |   |   |  |  |
|         | DTABLE  | 1     | YM  | 10 | L   |   |   |  |  |
|         | DRESP1  | 0     | 14  | 1  | 1   | 1 | 4 |  |  |
|         | DNODE   | 10    | 14  | 0  | 1   |   |   |  |  |
|         | DVPREL1 | 0     | 101 | 1  | 102 |   |   |  |  |

|  |         |    |     |    |     |    |     |  |  |
|--|---------|----|-----|----|-----|----|-----|--|--|
|  | DVCREL1 | 1  | 201 | 10 | 202 |    |     |  |  |
|  | DVMREL1 | 0  | 301 |    |     |    |     |  |  |
|  | DVPREL2 | 10 | 401 | 1  | 402 |    |     |  |  |
|  | DVCREL2 | 0  | 501 |    |     |    |     |  |  |
|  | DVMREL2 | 0  | 601 | 1  | 602 | 10 | 603 |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique identification number. (Integer > 0)  |
| LABEL      | User-defined label. (Character)  |
| EQID       | DEQATN entry identification number. (Integer > 0)  |
| FUNC       | Function to be applied to the arguments. See Remark 8. of the DRESP2 entry. (Character)  |
| REGION     | Region identifier for constraint screening. See Remark 5. of the DRESP2 entry. (Integer > 0)   |
| METHOD     | When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN) When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS) |
| Ci         | Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. of the DRESP2 entry. (Real; Defaults: C1 = 100., C2 = .005)   |
| “DESVAR”   | Flag indicating DESVAR entry identification numbers. (Character)   |
| DVSEIDI    | PART SE identification number for DVIDI (Integer $\geq 0$ )  |
| DVIDi      | DESVAR entry identification number. (Integer > 0)  |
| “DTABLE”   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)  |
| LBSEIDj    | PART SE identification number for LABLj. (Integer $\geq 0$ )   |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)   |
| “DRESP1”   | Flag indicating DRESP1 entry identification numbers. (Character)   |
| R1SEIDk    | PART SE identification number for NRk. (Integer $\geq 0$ )   |
| NRk        | DRESP1 entry identification number. (Integer > 0)  |
| “DNODE”    | Flag indicating grid point and component identification numbers. (Character)   |
| NDSEIDm    | PART SE identification number for (Gm,Cm). (Integer $\geq 0$ )   |
| Gm         | Identification number for any grid point in the model. (Integer > 0)   |
| Cm         | Component number of grid point Gm. (1 < Integer < 3)   |
| “DVPREL1”  | Flag indicating DVPREL1 entry identification number. (Character)   |

| Descriptor         | Meaning  |
|--------------------|--|
| P1SEIDI            | PART SE identification number for DPIP <sub>i</sub> . (Integer $\geq 0$ )  |
| DPIP <sub>i</sub>  | DVPREL1 entry identification number. (Integer $> 0$ )                      |
| “DVCREL1”          | Flag indicating DVCREL1 entry identification number. (Character)           |
| C1SEIDI            | PART SE identification number for DCIC <sub>i</sub> . (Integer $\geq 0$ )  |
| DCIC <sub>i</sub>  | DVCREL1 entry identification number. (Integer $> 0$ )                      |
| “DVMREL1”          | Flag indicating DVMREL2 entry identification number. (Character)           |
| M1SEIDI            | PART SE identification number for DMIM <sub>i</sub> . (Integer $\geq 0$ )  |
| DMIM <sub>i</sub>  | DVMREL1 entry identification number. (Integer $> 0$ )                      |
| “DVPREL2”          | Flag indicating DVPREL2 entry identification number. (Character)           |
| P2SEIDI            | PART SE identification number for DPI2P <sub>i</sub> . (Integer $\geq 0$ ) |
| DPI2P <sub>i</sub> | DVPREL2 entry identification number. (Integer $> 0$ )                      |
| “DVCREL2”          | Flag indicating DVCREL2 entry identification number. (Character)           |
| C2SEIDI            | PART SE identification number for DCI2C <sub>i</sub> . (Integer $\geq 0$ ) |
| DCI2C <sub>i</sub> | DVCREL2 entry identification number. (Integer $> 0$ )                      |
| “DVMREL2”          | Flag indicating DVMREL2 entry identification number. (Character)           |
| M2SEIDI            | PART SE identification number for DMI2M <sub>i</sub> . (Integer $\geq 0$ ) |
| DMI2M <sub>i</sub> | DVMREL2 entry identification number. (Integer $> 0$ )                      |

### Remarks:

1. SEDRSP2 is provided specifically for creating synthetic response with quantities from multiple PART SEs and must be placed in main bulk data section for consideration. SEDRSP2 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where seid>0.
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR, DVxRELY or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP2 follows rules and shares limitations of DRESP2. Rules and limitations of DRESP2 will not be repeated here.
4. SEDRSP2 does not support DRESP2 flag.

**SEDRSP3**

Defines External Response with User-Supplied Routines

Defines constituents from multiple PART SE for an external response using user-supplied routine(s).

**Format:**

| 1       | 2          | 3       | 4      | 5       | 6       | 7       | 8      | 9 | 10 |
|---------|------------|---------|--------|---------|---------|---------|--------|---|----|
| SEDRSP3 | ID         | LABEL   | GROUP  | TYPE    | REGION  |         |        |   |    |
|         | "DESVAR"   | DVSEID1 | DVID1  | DVSEID2 | DVID2   | DVSEID3 | DVID3  |   |    |
|         |            | DVSEID4 | DVID4  | -etc.-  |         |         |        |   |    |
|         | "DTABLE"   | LBSEID1 | LABL1  | LBSEID2 | LABL2   | LBSEID3 | LABL3  |   |    |
|         |            | LBSEID4 | LABL4  | -etc.-  |         |         |        |   |    |
|         | "DRESP1"   | R1SEID1 | NR1    | R1SEID2 | NR2     | R1SEID3 | NR3    |   |    |
|         |            | R1SEID4 | NR4    | -etc.-  |         |         |        |   |    |
|         | "DNODE"    | NDSEID1 | G1     | CMP1    | NDSEID2 | G2      | CMP2   |   |    |
|         |            | NDSEID3 | G3     | CMP3    | -etc.-  |         |        |   |    |
|         | "DVPREL1"  | P1SEID1 | DPIP1  | P1SEID2 | DPIP2   | P1SEID3 | DPIP3  |   |    |
|         |            | P1SEID4 | DPIP4  | -etc.-  |         |         |        |   |    |
|         | "DVCREL1"  | C1SEID1 | DCIC1  | C1SEID2 | DCIC2   | C1SEID3 | DCIC3  |   |    |
|         |            | C1SEID4 | DCIC4  | -etc.-  |         |         |        |   |    |
|         | "DVMREL1:" | M1SEID1 | DMIM1  | M1SEID2 | DMIM2   | M1SEID3 | DMIM3  |   |    |
|         |            | M1SEID4 | DMIM4  | -etc.-  |         |         |        |   |    |
|         | "DVPREL1"  | P2SEID  | DPI2P1 | P2SEID2 | DPI2P2  | P2SEID3 | DPI2P3 |   |    |
|         |            | P2SEID4 | DPI2P4 | -etc.-  |         |         |        |   |    |
|         | "DVCREL2"  | C2SEID1 | DC12C1 | C2SEID2 | DC12C2  | C2SEID3 | DC12C3 |   |    |
|         |            | C2SEID4 | DC12C4 | -etc.-  |         |         |        |   |    |
|         | "DVMREL2"  | M2SEID  | DMI2M1 | M2SEID2 | DMI2M2  | M2SEID3 | DMI2M3 |   |    |
|         |            | M2SEID4 | DMI2M4 | -etc.-  |         |         |        |   |    |
|         | "USRDATA"  |         |        |         | String  |         |        |   |    |
|         |            |         |        |         | -etc.-  |         |        |   |    |

**Example:**

|         |         |       |          |      |     |   |   |  |  |
|---------|---------|-------|----------|------|-----|---|---|--|--|
| SEDRSP3 | 10      | LBUCK | TAILWING | BUCK |     |   |   |  |  |
|         | DESVAR  | 0     | 101      | 1    | 3   | 1 | 4 |  |  |
|         |         | 10    | 201      |      |     |   |   |  |  |
|         | DTABLE  | 1     | YM       | 10   | L   |   |   |  |  |
|         | DRESP1  | 0     | 14       | 1    | 1   | 1 | 4 |  |  |
|         | DNODE   | 10    | 14       | 0    | 1   |   |   |  |  |
|         | DVPREL1 | 0     | 101      | 1    | 102 |   |   |  |  |

|  |         |                            |     |    |     |    |     |  |  |
|--|---------|----------------------------|-----|----|-----|----|-----|--|--|
|  | DVCREL1 | 1                          | 201 | 10 | 202 |    |     |  |  |
|  | DVMREL1 | 0                          | 301 |    |     |    |     |  |  |
|  | DVPREL2 | 10                         | 401 | 1  | 402 |    |     |  |  |
|  | DVCREL2 | 0                          | 501 |    |     |    |     |  |  |
|  | DVMREL2 | 0                          | 601 | 1  | 602 | 10 | 603 |  |  |
|  | USRDATA | Constants: 12345.67890.99. |     |    |     |    |     |  |  |

| Descriptor          | Meaning   |
|---------------------|---|
| ID                  | Unique identification number. (Integer > 0)   |
| LABEL               | User-defined label. (Character)   |
| GROUP               | Group name the external response type belongs to (Character). See Remark 2. of DRESP3.  |
| TYPE                | External response type (Character). See Remark 3. of the DRESP3.  |
| “DESVAR”            | Flag indicating DESVAR entry identification numbers. (Character)  |
| DVSEID <sub>i</sub> | PART SE identification number for DVID <sub>i</sub> . (Integer ≥ 0)   |
| DVID <sub>i</sub>   | DESVAR entry identification number. (Integer > 0)   |
| “DTABLE”            | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character) |
| LBSEID <sub>j</sub> | PART SE identification number for LABL <sub>j</sub> . (Integer ≥ 0)   |
| LABL <sub>i</sub>   | Label for a constant on the DTABLE or DTABLE2 entry. (Character)  |
| “DRESP1”            | Flag indicating DRESP1 entry identification numbers. (Character)  |
| R1SEID <sub>k</sub> | PART SE identification number for NR <sub>k</sub> . (Integer ≥ 0)   |
| NR <sub>k</sub>     | DRESP1 entry identification number. (Integer > 0)   |
| “DNODE”             | Flag indicating grid point and component identification numbers. (Character)  |
| NDSEID <sub>m</sub> | PART SE identification number for (G <sub>m</sub> ,C <sub>m</sub> ). (Integer ≥ 0)  |
| G <sub>m</sub>      | Identification number for any grid point in the model. (Integer > 0)  |
| C <sub>m</sub>      | Component number of grid point G <sub>m</sub> . (1 < Integer < 3)   |
| “DVPREL1”           | Flag indicating DVPREL1 entry identification number. (Character)  |
| P1SEID <sub>i</sub> | PART SE identification number for DPIP <sub>i</sub> . (Integer ≥ 0)   |
| DPIP <sub>i</sub>   | DVPREL1 entry identification number. (Integer > 0)  |
| “DVCREL1”           | Flag indicating DVCREL1 entry identification number. (Character)  |
| C1SEID <sub>i</sub> | PART SE identification number for DCIC <sub>i</sub> . (Integer ≥ 0)   |
| DCIC <sub>i</sub>   | DVCREL1 entry identification number. (Integer > 0)  |
| “DVMREL1”           | Flag indicating DVMREL2 entry identification number. (Character)  |
| M1SEID <sub>i</sub> | PART SE identification number for DMIM <sub>i</sub> . (Integer ≥ 0)   |

| Descriptor | Meaning  |
|------------|--|
| DMIMi      | DVMREL1 entry identification number. (Integer > 0)               |
| “DVPREL2”  | Flag indicating DVPREL2 entry identification number. (Character) |
| P2SEIDi    | PART SE identification number for DPI2Pi. (Integer $\geq 0$ )    |
| DPI2Pi     | DVPREL2 entry identification number. (Integer > 0)               |
| “DVCREL2”  | Flag indicating DVCREL2 entry identification number. (Character) |
| C2SEIDi    | PART SE identification number for DCI2Ci. (Integer $\geq 0$ )    |
| DCI2Ci     | DVCREL2 entry identification number. (Integer > 0)               |
| “DVMREL2”  | Flag indicating DVMREL2 entry identification number. (Character) |
| M2SEIDi    | PART SE identification number for DMI2Mi. (Integer $\geq 0$ )    |
| DMI2Mi     | DVMREL2 entry identification number. (Integer > 0)               |
| “USRDATA”  | Flag indicating user input data. (Character).                    |

**Remarks:**

1. SEDRSP3 is provided specifically for creating external response with quantities from multiple PART SEs and must be placed in main Bulk Data Section for consideration. SEDRSP3 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where seid>0.
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR, DVxRELY or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP3 follows rules and shares limitations of DRESP3. Rules and limitations of DRESP3 will not be repeated here.
4. SEDRSP3 does not support DRESP2 flag.

**SEELT****Superelement Boundary Element Reassignment**

Reassigns superelement boundary elements to an upstream superelement.

**Format:**

| 1     | 2    | 3    | 4      | 5    | 6    | 7    | 8    | 9    | 10 |
|-------|------|------|--------|------|------|------|------|------|----|
| SEELT | SEID | EID1 | EID2   | EID3 | EID4 | EID5 | EID6 | EID7 |    |
|       | EID8 | EID9 | -etc.- |      |      |      |      |      |    |

**Example:**

|       |   |     |     |     |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|
| SEELT | 2 | 147 | 562 | 937 |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|

**Alternate Format and Example:**

|       |      |       |        |       |  |  |  |  |  |
|-------|------|-------|--------|-------|--|--|--|--|--|
| SEELT | SEID | EID1  | "THRU" | EID2  |  |  |  |  |  |
| SEELT | 5    | 12006 | THRU   | 12050 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. See Remark 7. (Integer > 0)                         |
| EIDI       | Element identification numbers. (Integer > 0 or "THRU"; for "THRU" option EID1 < EID2.) |

**Remarks:**

- Elements connected entirely to the exterior points of an upstream superelement are called boundary elements and are assigned to the downstream superelement. The SEELT entry provides the means of reassigning the element to the upstream superelement. This entry may be applied to boundary elements only.
- Open sets are allowed with the "THRU" option.
- Elements processed with primary superelements will also be contained in any referencing secondary superelement.
- EIDI may refer to plot elements, general elements, and structural elements.
- This entry does not change the exterior grid point set of the superelement.
- SEELT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
- If the model contains a BEGIN SUPER, the SEELT entry will assign the specified elements to the SEID (not just the boundary elements). This is an alternative to using SESET. In this case, SEID = 0 is a valid entry. For further information, refer to the [MSC Nastran Reference Manual](#).

**SEECLD** Partitioned Superelement Exclusion

Defines grids that will be excluded during the attachment of a partitioned superelement.

**Format:**

| 1       | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| SEEXCLD | SEIDA | SEIDB | GIDA1  | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | -etc.- |       |       |       |       |       |    |

**Example1:**

|         |     |    |    |     |     |  |  |  |  |
|---------|-----|----|----|-----|-----|--|--|--|--|
| SEEXCLD | 110 | 10 | 45 | 678 | 396 |  |  |  |  |
|---------|-----|----|----|-----|-----|--|--|--|--|

**Example 2:**

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| SEEXCLD | 400 | ALL | 10 | 20 | 30 | THRU | 40 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SEIDA      | Partitioned superelement identification number. (Integer $\geq 0$ )   |
| SEIDB      | Superelement identification. (Integer $\geq 0$ or Character = "ALL"; Default = "ALL")   |
| GIDAi      | Identification number of a grid in superelement SEIDA to be excluded from connection to superelement SEIDB. (Integer $> 0$ or "THRU"; for "THRU" option GIDA1 < GIDA2.) |

**Remarks:**

1. SEECLD can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. SEIDA and SEIDB may reference only substructures or the residual structure, that is, parts defined after a BEGIN SUPER = entry.
3. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SELABEL** Superelement Output Label

Defines a label or name to be printed in the superelement output headings.

**Format:**

| 1       | 2    | 3 | 4 | 5     | 6 | 7 | 8 | 9 | 10 |
|---------|------|---|---|-------|---|---|---|---|----|
| SELABEL | SEID |   |   | LABEL |   |   |   |   |    |

**Example:**

|         |    |                                 |  |
|---------|----|---------------------------------|--|
| SELABEL | 10 | LEFT REAR FENDER, MODEL XYZ2000 |  |
|---------|----|---------------------------------|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned superelement identification number. (Integer > 0)            |
| LABEL      | Label associated with superelement SEID for output headings. (Character) |

**Remarks:**

1. SELABEL can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Only one SELABEL per superelement may be specified.
3. The label will appear in all superelement output headings. However, in some headings the label may be truncated.
4. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SELOC****Partitioned Superelement Location**

Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.

**Format:**

| 1     | 2    | 3   | 4   | 5   | 6   | 7   | 8   | 9 | 10 |
|-------|------|-----|-----|-----|-----|-----|-----|---|----|
| SELOC | SEID | PA1 | PA2 | PA3 | PB1 | PB2 | PB3 |   |    |

**Example:**

|       |     |    |     |     |      |     |    |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|
| SELOC | 110 | 10 | 100 | 111 | 1010 | 112 | 30 |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned identification number of the partitioned superelement. (Integer > 0)   |
| PAi        | Identification numbers of three noncolinear grids (GRID entry) or points (POINT entry) which are in the partitioned superelement. (Integer > 0)              |
| PBi        | Identification numbers of three grids (GRID entry) or points (POINT entry) defined in the main Bulk Data Section to which PAi will be aligned. (Integer > 0) |

**Remarks:**

1. SELOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. The superelement will be rotated and translated for alignment of the GAi and GBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in superelement SEID.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section. If they belong to a superelement that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three grids or points are required even if the superelement connects to only one or two exterior grids.
8. Coordinate systems, global displacement directions, and element coordinate systems for the superelement will rotated and translated.
9. The global coordinate directions of the boundary grid points of the upstream superelement will be transformed internally to the global coordinate directions of the attachment grid points in the downstream superelement. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the superelement to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:

- The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
  - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
  - The z-axis is defined by the cross product of the x-axis into the y-axis.
  - The rotation and translation transformation aligns the local system defined by the superelement grids with the local system defined by the main Bulk Data Section grids.
11. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SEMPLN****Superelement Mirror Plane**

Defines a mirror plane for mirroring a partitioned superelement.

**Format:**

| 1      | 2    | 3       | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|--------|------|---------|----|----|----|---|---|---|----|
| SEMPLN | SEID | "PLANE" | P1 | P2 | P3 |   |   |   |    |

**Example:**

|        |     |       |    |    |      |  |  |  |  |
|--------|-----|-------|----|----|------|--|--|--|--|
| SEMPLN | 110 | PLANE | 12 | 45 | 1125 |  |  |  |  |
|--------|-----|-------|----|----|------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned superelement identification number. (Integer > 0).                         |
| "PLANE"    | Flag indicating that the plane is defined by three noncolinear points.                 |
| Pi         | GRID or POINT entry identification numbers of three noncolinear points. (Integer > 0). |

**Remarks:**

1. SEMPLN can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. Grids or points referenced on this entry must be defined in the main Bulk Data Section.

**SENQSET**

Superelement Internal Generalized Degree-of-Freedom

Defines number of internally generated scalar points for superelement dynamic reduction.

**Format:**

| 1       | 2    | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|---|---|---|---|---|---|---|----|
| SENQSET | SEID | N |   |   |   |   |   |   |    |

**Example:**

|         |     |    |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|
| SENQSET | 110 | 45 |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned superelement identification number. See Remark 3. (Integer > 0 or Character = "ALL")                       |
| N          | Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer > 0; Default = 0) |

**Remarks:**

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID = "ALL" will automatically generate N q-set degrees-of-freedom for all superelements, except the residual structure (SEID = 0). Specifying additional SENQSET entries for specific superelements will override the value of N specified on this entry.
4. If the user manually specifies q-set degrees-of-freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.
5. See PARAM,NQSET for an alternate method of specifying QSET degree-of-freedoms.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SEQGP****Grid and Scalar Point Resequencing**

Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

**Format:**

| 1     | 2   | 3      | 4   | 5      | 6   | 7      | 8   | 9      | 10 |
|-------|-----|--------|-----|--------|-----|--------|-----|--------|----|
| SEQGP | ID1 | SEQID1 | ID2 | SEQID2 | ID3 | SEQID3 | ID4 | SEQID4 |    |

**Example:**

|       |      |      |     |     |   |     |   |   |  |
|-------|------|------|-----|-----|---|-----|---|---|--|
| SEQGP | 5392 | 15.6 | 596 | 0.2 | 2 | 1.9 | 3 | 2 |  |
|-------|------|------|-----|-----|---|-----|---|---|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)    |
| SEQIDi     | Sequenced identification number. (Real > 0.0 or Integer > 0) |

**Remarks:**

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID. If the SEQID is real and < 1.0 the value must be entered as 0.2 not .2 (see field 5 of example).
2. The SEQIDi numbers must be unique and may not be the same as a point IDi which is not being changed. No grid point IDi may be referenced more than once.
3. From one to four grid or scalar points may be resequenced on a single entry.
4. If a point IDi is referenced more than once, the last reference will determine its sequence.
5. Automatic resequencing is also available. See [OLDSEQ, 963](#).

**SEQSEP****Superelement Sequences**

Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.

**Format:**

| 1      | 2    | 3    | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|------|------|--------|-----|-----|-----|-----|-----|----|
| SEQSEP | SSID | PSID | GP1    | GP2 | GP3 | GP4 | GP5 | GP6 |    |
|        | GP7  | GP8  | -etc.- |     |     |     |     |     |    |

**Example:**

|        |     |    |     |     |     |     |  |  |  |
|--------|-----|----|-----|-----|-----|-----|--|--|--|
| SEQSEP | 121 | 21 | 109 | 114 | 124 | 131 |  |  |  |
|--------|-----|----|-----|-----|-----|-----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SSID       | Identification number for secondary superelement. (Integer $> 0$ ).                        |
| PSID       | Identification number for the primary superelement. (Integer $\geq 0$ ).                   |
| GPi        | Exterior grid point identification numbers for the primary superelement. (Integer $> 0$ ). |

**Remarks:**

1. This entry is not needed if the grid points listed on the CSUPER entry with the same SSID are in the order of the corresponding exterior grid points of the primary superelement.
2. In [Figure 9-154](#), the exterior grid points of 10, 20, and 30 of SEID = 1 correspond to the points 13, 12, and 11, respectively, of image SEID = 2. The CSUPER entry may be defined alone or with a SEQSEP entry as shown in [Figure 9-154](#).

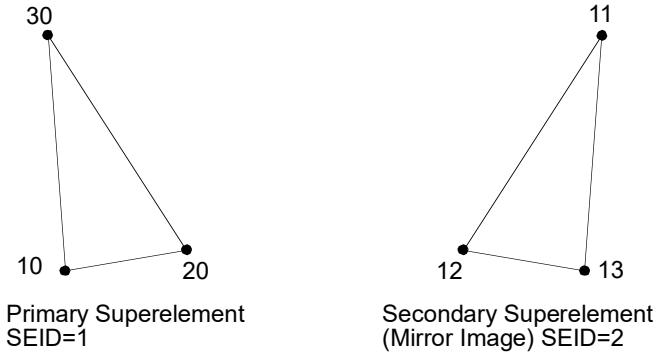


Figure 9-154 Grid Point Correspondence Between Primary and Secondary Superelements

**CSUPER Entry Only:**

| 1      | 2 | 3 | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|--------|---|---|----|----|----|---|---|---|----|
| CSUPER | 2 | 1 | 13 | 12 | 11 |   |   |   |    |

**CSUPER and SEQSEP Entries:**

|        |   |   |    |    |    |  |  |  |  |
|--------|---|---|----|----|----|--|--|--|--|
| CSUPER | 2 | 1 | 11 | 12 | 13 |  |  |  |  |
| SEQSEP | 2 | 1 | 30 | 20 | 10 |  |  |  |  |

**SEQSET**

## Superelement Generalized Degree-of-Freedom

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

**Format:**

| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|--------|------|-----|----|-----|----|-----|----|---|----|
| SEQSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|        |    |   |        |   |   |    |   |  |  |
|--------|----|---|--------|---|---|----|---|--|--|
| SEQSET | 15 | 1 | 123456 | 7 | 5 | 22 | 3 |  |  |
|--------|----|---|--------|---|---|----|---|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. Must be a primary superelement. (Integer > 0)  |
| Ci         | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points.) |
| IDi        | Grid or scalar point identification numbers. Must be an exterior point. (Integer > 0)  |

**Remarks:**

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these degrees-of-freedom for secondary superelements.

**SEQSET1**

## Superelement Generalized Degree-of-Freedom, Alternate Form

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

**Format:**

| 1       | 2    | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|--------|----|----|----|----|----|----|
| SEQSET1 | SEID | C  | G1     | G2 | G3 | G4 | GS | G6 |    |
|         | G7   | G8 | -etc.- |    |    |    |    |    |    |

**Example:**

|         |    |        |   |   |   |    |     |   |  |
|---------|----|--------|---|---|---|----|-----|---|--|
| SEQSET1 | 15 | 123456 | 1 | 7 | 9 | 22 | 105 | 6 |  |
|         | 52 | 53     |   |   |   |    |     |   |  |

**Alternate Format and Example:**

|         |      |   |     |        |     |  |  |  |  |
|---------|------|---|-----|--------|-----|--|--|--|--|
| SEQSET1 | SEID | C | G1  | "THRU" | G2  |  |  |  |  |
| SEQSET1 | 16   | 0 | 101 | THRU   | 110 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. Must be a primary superelement. (Integer > 0)  |
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi         | Grid or scalar point identification numbers. Must be exterior points. (Integer > 0 or "THRU"; for THRU option G1 < G2.)                                |

**Remarks:**

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set, to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these entries for secondary superelements.

**SESET**

## Superelement Interior Point Definition

Defines interior grid points for a superelement.

**Format:**

| 1     | 2    | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|------|----|----|----|----|----|----|----|----|
| SESET | SEID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

**Example:**

|       |   |   |    |    |    |     |  |  |  |
|-------|---|---|----|----|----|-----|--|--|--|
| SESET | 5 | 2 | 17 | 24 | 25 | 165 |  |  |  |
|-------|---|---|----|----|----|-----|--|--|--|

**Alternate Format and Example:**

|       |      |    |        |     |  |  |  |  |  |
|-------|------|----|--------|-----|--|--|--|--|--|
| SESET | SEID | G1 | "THRU" | G2  |  |  |  |  |  |
| SESET | 2    | 17 | THRU   | 165 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. Must be a primary superelement. (Integer $\geq 0$ )   |
| Gi         | Grid or scalar point identification number. ( $0 < \text{Integer} < 100000000$ ; G1 < G2) |

**Remarks:**

1. Interior grid points may also be defined via field 9 of the GRID and GRIDG Bulk Data entries. The SESET entry takes precedence over the SEID field on the GRID or GRIDG entries. SESET defines grid and scalar points to be included as interior to a superelement. SESET may be used as the primary means of defining superelements or it may be used in combination with SEELT entries which define elements interior to a superelement.
2. Gi may appear on an SESET entry only once.
3. Scalar points are ignored unless a BEGIN SUPER is in the file.
4. Open sets are allowed with the "THRU" option. Missing grid points (whether in "THRU" range or mentioned explicitly) are not identified.
5. All degrees-of-freedom for Gi are placed in the o-set of the superelement. See [Degree-of-Freedom Sets, 1111](#).
6. SESET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.

**SESUP**

## Fictitious Support

Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

**Format:**

| 1     | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|-------|------|-----|----|-----|----|-----|----|---|----|
| SESUP | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|       |   |    |     |  |  |  |  |  |  |
|-------|---|----|-----|--|--|--|--|--|--|
| SESUP | 5 | 16 | 215 |  |  |  |  |  |  |
|-------|---|----|-----|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SEID       | Superelement identification number. Must a primary superelement. (Integer > 0)  |
| IDi        | Grid or scalar point identification number. Must be exterior points. (Integer > 0)  |
| Ci         | Component numbers. (Integer zero or blank for scalar points; Any unique combination of the Integers 1 through 6 for grid points.) |

**Remarks:**

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. The Ci degrees-of-freedom must be exterior degrees-of-freedom of the SEID superelement.
3. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
4. There must be a sufficient number of degrees-of-freedom on SESUP entries to discard any free body modes of the superelement.
5. SESUP Bulk Data entries are not allowed for part (partitioned bulk data) superelements. Use the SUPPORT Bulk Data records to identify component rigid body modes.

**SET1****Set Definition**

Defines a list of structural grid points or element identification numbers.

**Format:**

| 1    | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| SET1 | SID | ID1    | ID2 | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|      | ID8 | -etc.- |     |     |     |     |     |     |    |

**Example 1:**

|      |    |    |    |    |     |    |    |    |  |
|------|----|----|----|----|-----|----|----|----|--|
| SET1 | 3  | 31 | 62 | 93 | 124 | 16 | 17 | 18 |  |
|      | 19 |    |    |    |     |    |    |    |  |

**Example 2:**

|      |    |    |    |      |    |    |      |    |  |
|------|----|----|----|------|----|----|------|----|--|
| SET1 | 6  | 29 | 32 | THRU | 50 | 61 | THRU | 70 |  |
|      | 17 | 57 |    |      |    |    |      |    |  |

**Example 3:**

|      |   |      |  |  |  |  |  |  |  |
|------|---|------|--|--|--|--|--|--|--|
| SET1 | 7 | SKIN |  |  |  |  |  |  |  |
|------|---|------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique identification number. (Integer > 0)  |
| IDi        | List of structural grid point or element identification numbers. (Integer > 0 or “THRU”; for the “THRU” option, ID1 < ID2 or “SKIN”; in field 3) |

**Remarks:**

- When using the “THRU” option for SPLINEi or PANEL data entries, all intermediate grid points must exist.
- When using the “THRU” option for XYOUTPUT or AECOMP requests, missing grid points are ignored. The first and last points must exist.
- When using the “SKIN” option, a panel will be generated consisting of the structural portion of the fluid-structural boundary. This option works ONLY with all fields of ACMODL having default value.
- THRU may not appear in field 3 or 9 (2 or 9 for continuations).
- RC network solver does not support SET1 for thermal analysis.

**SET2**

## Grid Point List

Defines a list of structural grid points in terms of aerodynamic macro elements.

**Format:**

| 1    | 2   | 3     | 4   | 5   | 6   | 7   | 8    | 9    | 10 |
|------|-----|-------|-----|-----|-----|-----|------|------|----|
| SET2 | SID | MACRO | SP1 | SP2 | CH1 | CH2 | ZMAX | ZMIN |    |

**Example:**

|      |   |     |     |      |     |       |      |  |  |
|------|---|-----|-----|------|-----|-------|------|--|--|
| SET2 | 3 | 111 | 0.0 | 0.75 | 0.0 | 0.667 | 3.51 |  |  |
|------|---|-----|-----|------|-----|-------|------|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique identification number. (Integer > 0)   |
| MACRO      | Element identification number of an aerodynamic macro element. (Integer > 0)  |
| SP1, SP2   | Lower and higher span division points defining the prism containing the set. (Real)   |
| CH1, CH2   | Lower and higher chord division points defining the prism containing the set. (Real)  |
| ZMAX, ZMIN | Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a CAEROi entry) of the prism containing set. (Real) |

**Remarks:**

1. The SET2 entry is referenced by the SPLINEi entry.
2. Every grid point within the defined prism and within the height range will be in the list. For example:

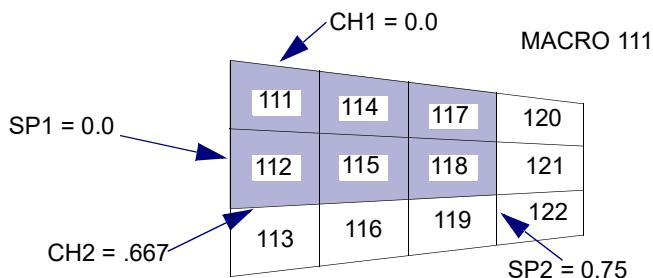


Figure 9-155 SET2 Entry Example.

The shaded area in Figure 9-155 defines the cross section of the prism for the sample data given above. Points exactly on the boundary may be missed; therefore, to get all the grid points within the area of the macro element,  $SP1=-.01$ ,  $SP2=1.01$ , etc. should be used.

3. A zero value for ZMAX or ZMIN implies a value of infinity. Usually,  $ZMAX \geq 0.0$  and  $ZMIN \leq 0.0$ .
4. To print the (internal) grid IDs found, use DIAG 18.

**SET3****Labeled Set Definition**

Defines a list of grids, elements or points.

**Format:**

| 1    | 2   | 3   | 4     | 5   | 6   | 7   | 8   | 9   | 10 |
|------|-----|-----|-------|-----|-----|-----|-----|-----|----|
| SET3 | SID | DES | ID1   | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|      | ID7 | ID8 | -etc- |     |     |     |     |     |    |

**Example:**

|      |   |       |    |    |    |    |    |    |  |
|------|---|-------|----|----|----|----|----|----|--|
| SET3 | 1 | POINT | 11 | 12 | 13 | 15 | 18 | 21 |  |
|------|---|-------|----|----|----|----|----|----|--|

**Alternate Format and Example:**

|      |     |       |     |        |     |  |  |  |  |
|------|-----|-------|-----|--------|-----|--|--|--|--|
| SET3 | SID | DES   | ID1 | "THRU" | ID2 |  |  |  |  |
| SET3 | 33  | POINT | 20  | THRU   | 60  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique identification number. (Integer>0)  |
| DES        | Set description (Character). Valid options are "GRID", "ELEM", "POINT", "PROP", "RBEin", and "RBEx". |
| IDi        | Identifiers of grids points, elements, points or properties. (Integer > 0)                           |

**Remarks:**

1. If a SET3 entry is referenced on a PBMSECT or PBRSECT entry, the POINTs must lie in the (xy) plane of the basic coordinate system, and be in the order when traversing the boundary or the profile.
2. When the SET3 entry is referenced by a panel, describers can be "GRID", "ELEM" or "PROP".
3. THRU may not appear in field 4 or 9 (2 or 9 for continuations).
4. When SET3 is referenced by SOLs 400 entry only GRID or ELEM may be used.
5. When SET3 is referenced by RFORCE (IDRF field) for SOL 600, only ELEM may be used.
6. When SET3 is referenced by SOL 400 DEACTEL entries, only ELEM may be used.
7. When a SET3 is referenced by a ELSIDI or XELSIDi field on an FTGDEF entry, only ELEM may be used. When SET3 is referenced by a NDSIDI field on a FTGDEF entry, only GRID may be used.
8. When DES="RBEin", the SET selects rigid elements to be included for MPC=sid and is applicable to Rigid Element types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1. Note that Rigid Elements with duplicate ID across Rigid Element types will all be utilized.

9. For DES="RBEex", the SET selects rigid elements to be excluded for MPC=sid and is applicable to Rigid Elements types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1.
10. Note that "RBEin" and "RBEex" are mutually exclusive and should not appear together for a single SET.
11. By default, without SET3,mpcid,RBExx, all Rigid Elements in the input deck will be used.
12. SET selection for rigid elements does not cover additional IDs on MPCADD bulk data entry. This kind of selection is not supported in SOL 106 either.
13. Rigid element set section is supported in SOL 400 if 'RIGID=LINEAR' is present in case control deck.

**SET4****Property Set Definition**

Defines a list of property IDs

**Format:**

| 1    | 2   | 3     | 4    | 5     | 6   | 7   | 8   | 9   | 10 |
|------|-----|-------|------|-------|-----|-----|-----|-----|----|
| SET4 | ID  | CLASS | TYPE | ID1   | ID2 | ID3 | ID4 | ID5 |    |
|      | ID6 | ID7   | ID8  | -etc- |     |     |     |     |    |

**Example:**

|      |    |      |        |   |      |    |  |  |  |
|------|----|------|--------|---|------|----|--|--|--|
| SET4 | 22 | PROP | PSOLID | 1 | THRU | 20 |  |  |  |
|------|----|------|--------|---|------|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number. (Integer>0).                                      |
| CLASS      | Set to "PROP". (Character = PROP; no default)                                   |
| TYPE       | Property type. Valid options are PSOLID, PHELL, PSHEAR, PBAR, PBEAM, and PWELD. |
| IDi        | Property IDs of the specified TYPE flag.  |

**Remarks:**

1. THRU option may not appear in field 5 or 9 on first line or field 2 or 9 for continuation lines.
2. Currently referenced from the FTGDEF entry.

**SETREE**

Superelement Tree Definition (Alternate Form of DTI,SETREE)

Specifies superelement reduction order.

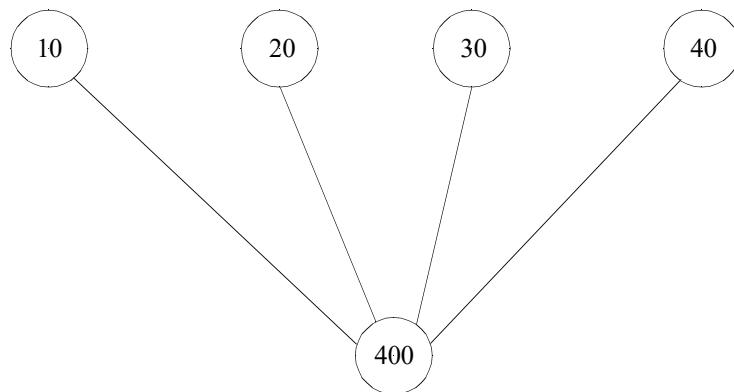
**Format:**

| 1      | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|--------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| SETREE | SEID  | SEUP1 | SEUP2  | SEUP3 | SEUP4 | SEUP5 | SEUP6 | SEUP7 |    |
|        | SEUP8 | SEUP9 | -etc.- |       |       |       |       |       |    |

**Example:**

|        |     |    |    |    |    |  |  |  |  |
|--------|-----|----|----|----|----|--|--|--|--|
| SETREE | 400 | 10 | 20 | 30 | 40 |  |  |  |  |
|--------|-----|----|----|----|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Partitioned superelement identification number of a downstream superelement. (Integer $\geq 0$ ) |
| SEUPI      | Identification number of superelements that are upstream of SEID. (Integer $> 0$ )               |

**Remarks:**

1. SETREE entries or DTI,SETREE entry are required for multilevel superelement configurations.
2. At least one SETREE entry is required for each nontip superelement, including the residual structure (SEID = 0). Multiple SETREE entries with the same SEID are allowed.
3. A superelement may appear only once in an SEUPI field on all SETREE entries.
4. If an DTI,SETREE entry is provided, then SETREE entries are not required.
5. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.

6. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one or more of the superelements are not in the same path.
7. The SETREE entry will only work if PART (BEGIN SUPER) superelements exist in the model. If there are no PARTs in the model, the SETREE entries will be ignored.

**SEUSET**

## Superelement Degree-of-Freedom Set Definition

Defines a degree-of-freedom set for a superelement.

**Format:**

| 1      | 2    | 3     | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|--------|------|-------|-----|----|-----|----|-----|----|----|
| SEUSET | SEID | SNAME | ID1 | C1 | ID2 | C2 | ID3 | C3 |    |

**Example:**

|        |    |    |   |        |   |   |    |   |  |
|--------|----|----|---|--------|---|---|----|---|--|
| SEUSET | 15 | U4 | 1 | 123456 | 7 | 5 | 22 | 3 |  |
|--------|----|----|---|--------|---|---|----|---|--|

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. (Integer > 0)  |
| SNAME      | Set name. (One to four characters or string “ZERO”, followed by the set name.)   |
| IDi        | Grid or scalar point identification numbers. (Integer > 0)   |
| Ci         | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blank for grid points; Integer 0 or blank for scalar points.) |

**Remarks:**

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names “U1” through “U6” or their new names on the DEFUSET entry.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

**SEUSET1**

## Superelement Degree-of-Freedom Set Definition, Alternate Form

Defines a degree-of-freedom set for a superelement.

**Format:**

| 1       | 2    | 3     | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|-------|--------|----|----|----|----|----|----|
| SEUSET1 | SEID | SNAME | C      | G1 | G2 | G3 | G4 | G5 |    |
|         | G6   | G7    | -etc.- |    |    |    |    |    |    |

**Example:**

|         |    |    |   |    |    |    |    |    |  |
|---------|----|----|---|----|----|----|----|----|--|
| SEUSET1 | 15 | U4 | 1 | 12 | 15 | 17 | 22 | 25 |  |
|         | 52 | 53 |   |    |    |    |    |    |  |

**Alternate Format and Example:**

|         |      |       |   |    |        |    |  |  |  |
|---------|------|-------|---|----|--------|----|--|--|--|
| SEUSET1 | SEID | SNAME | C | G1 | "THRU" | G2 |  |  |  |
| SEUSET1 | 15   | U4    | 1 | 12 | THRU   | 27 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SEID       | Superelement identification number. (Integer > 0)  |
| SNAME      | Set name. (One to four characters or string "ZERO", followed by the set name.)   |
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi         | Grid or scalar point identification number. (Integer > 0)  |

**Remarks:**

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names "U1" through "U6" or their new names on the DEFUSET entry.
2. If SNAME= "ZERO*i*", where *i* is a set name, then the degrees-of-freedom are omitted from set *i*.
3. If the alternate format is used, all of the points G1 through G2 are assigned to the set.

**SHREL****Elastic Shear Model**

Defines an elastic shear model with a constant shear modulus. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|---|---|---|---|---|---|----|
| SHREL | SID | G |   |   |   |   |   |   |    |

**Example:**

|       |     |       |  |  |  |  |  |  |  |
|-------|-----|-------|--|--|--|--|--|--|--|
| SHREL | 250 | 80.E6 |  |  |  |  |  |  |  |
|-------|-----|-------|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique shear model number referenced from a MATDEUL entry. (Integer > 0;<br>Required) |
| G          | Shear-modulus value. (Real; Default = 0.0)  |

**Remark:**

1. SID must unique among all SHRxx entries in one model.

**SHRPOL****Polynomial Shear Model**

Defines an elastic shear model with a polynomial shear modulus. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|-------|-------|---|---|---|----|
| SHRPOL | SID | $G_0$ | $G_1$ | $G_2$ | $G_3$ |   |   |   |    |

**Example:**

|        |     |       |  |  |  |  |  |  |  |
|--------|-----|-------|--|--|--|--|--|--|--|
| SHRPOL | 250 | 80.E6 |  |  |  |  |  |  |  |
|--------|-----|-------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique shear model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| $G_0$      | Coefficient $G_0$ . (Real; Default = 0.0)  |
| $G_1$      | Coefficient $G_1$ . (Real; Default = 0.0)  |
| $G_2$      | Coefficient $G_2$ . (Real; Default = 0.0)  |
| $G_3$      | Coefficient $G_3$ . (Real; Default = 0.0)  |

**Remark:**

1. SID must unique among all SHRx entries in one model.
2. The shear modulus is computed from

$$G = G_0 + G_1\gamma + G_2\gamma^2 + G_3\gamma^3$$

where  $\gamma$  = effective plastic shear strain

and  $G_0$ ,  $G_1$ ,  $G_2$  and  $G_3$  are constants

**SHRUDS**

User-defined Shear Model for Elements.

Specifies that a user subroutine is being used to define the shear modulus. Use in SOL700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|---|---|---|---|---|----|
| SHRUDS | SID | GROUP | UNAME |   |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE mymat 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| SHRUDS | 12 | mymat | EXSHR |   |   |   |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique porosity model ID. (Integer > 0; Required)   |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement.<br>(Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXSHR)                    |

**Remarks:**

1. Only can be used for SOL 700.
2. The SID must be referenced by a MATDEUL or MAT1 entry.
3. UNAME=EXSHR can only be used.

**SLBDY**

## Slot Boundary List

**Format:**

| 1     | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| SLBDY | RHO | M      | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|       | ID7 | -etc.- |     |     |     |     |     |     |    |

**Example:**

|       |       |   |    |    |    |    |    |    |  |
|-------|-------|---|----|----|----|----|----|----|--|
| SLBDY | 0.002 | 6 | 16 | 17 | 18 | 25 | 20 | 21 |  |
|       | 22    |   |    |    |    |    |    |    |  |

| Descriptor | Meaning   |
|------------|---|
| RHO        | Density of fluid at boundary. (Real > 0.0 or blank)   |
| M          | Number of slots. (Integer $\geq 0$ or blank)  |
| IDj        | Identification numbers of GRIDS slot points at boundary with axisymmetric fluid cavity, j = 1, 2, ..., J. (Integer > 0) |

**Remarks:**

1. SLBDY is allowed only if an AXSLOT entry is also present.
2. If RHO or M is blank, the default value on the AXSLOT entry is used. The effective value must not be zero for RHO. If the effective value of M is zero, no matrices at the boundary will be generated.
3. The order of the list of points determines the topology of the boundary. The points are listed sequentially as one travels along the boundary in either direction. At least two points must be defined.

**SLOAD**

## Static Scalar Load

Defines concentrated static loads on scalar or grid points.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-------|-----|----|----|----|----|----|----|---|----|
| SLOAD | SID | S1 | F1 | S2 | F2 | S3 | F3 |   |    |

**Example:**

|       |    |   |     |    |      |    |       |  |  |
|-------|----|---|-----|----|------|----|-------|--|--|
| SLOAD | 16 | 2 | 5.9 | 17 | -6.3 | 14 | -2.93 |  |  |
|-------|----|---|-----|----|------|----|-------|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)             |
| Si         | Scalar or grid point identification number. (Integer > 0) |
| Fi         | Load magnitude. (Real)                                    |

**Remarks:**

1. In the static solution sequences, SID is selected by the LOAD Case Control command.
2. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
3. Up to three loads may be defined on a single entry.
4. If Si refers to a grid point, the load is applied to component T1 of the displacement coordinate system (see the CD field on the GRID entry).

**SLOADN1**

Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400

Defines concentrated static loads on grid points of heat shell elements with linear or quadratic temperature distribution through the thickness direction.

**Format:**

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|---------|-----|----|----|----|----|----|----|---|----|
| SLOADN1 | SID | G1 | C1 | Q1 | G2 | C2 | Q2 |   |    |

**Example:**

|         |    |    |    |       |    |   |       |  |  |
|---------|----|----|----|-------|----|---|-------|--|--|
| SLOADN1 | 10 | 10 | 12 | 1300. | 20 | 2 | 1300. |  |  |
|---------|----|----|----|-------|----|---|-------|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Load set identification number. (Integer > 0)   |
| Gi         | Grid point identification number. (Integer > 0)   |
| Ci         | Composite numbers. ( $0 \leq \text{Integer} \leq 3$ ; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1) |
| Qi         | Power. (Real)   |

**Remarks:**

1. This entry is for shell elements defined on a PSHLN1 heat transfer to specify power input.
2. In the steady-state heat transfer analysis, SID is selected by the LOAD Case Control command.
3. In the transient heat transfer analysis, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.

**SNORM**

## Surface Normal Vector at Grid Point

Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

**Format:**

| 1     | 2   | 3   | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|-------|-----|-----|----|----|----|---|---|---|----|
| SNORM | GID | CID | N1 | N2 | N3 |   |   |   |    |

**Example:**

|       |   |   |    |     |    |  |  |  |  |
|-------|---|---|----|-----|----|--|--|--|--|
| SNORM | 3 | 2 | 0. | -1. | 0. |  |  |  |  |
|-------|---|---|----|-----|----|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| GID        | Unique grid point identification number. (Integer > 0)  |
| CID        | Identification number of coordinate system in which the components of the normal vector are defined. See Remark 3. (Integer $\geq 0$ ; Default = 0 for the basic coordinate system) |
| Ni         | Components of normal vector. The three components of the normal need not define a unit vector. (Real; Default = 0.0)  |

**Remarks:**

1. The SNORM Bulk Data entry overrides any unique, internally-generated grid point normals that may have been requested with the user parameter SNORM, described in Chapter 6 of this guide.
2. The normal is used in CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. For all other elements, the normal is ignored.
3. If CID is a cylindrical or spherical coordinate system, the components Ni are in the local tangent system at grid GID. For example, if CID=10 is a spherical coordinate system and normals must be defined pointing outwards in the radial direction of the sphere, see , then the SNORM entries for all grids GID on the sphere are simply  
SNORM, GID, 10, 1., 0., 0.

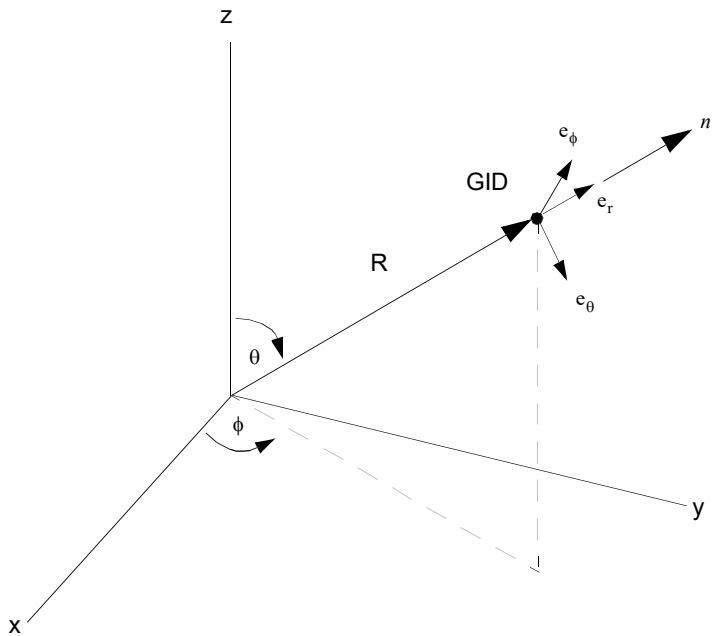


Figure 9-156

**SPBLND1**

## Strip Based Spline Blending

Defines a strip based blending of two splines.

**Format:**

| 1       | 2   | 3    | 4    | 5   | 6  | 7   | 8  | 9  | 10 |
|---------|-----|------|------|-----|----|-----|----|----|----|
| SPBLND1 | SID | SID1 | SID2 | OPT | W1 | GID | D1 | D2 |    |
|         | X1  | X2   | X3   | CID |    |     |    |    |    |

**Example:**

|         |     |     |     |     |  |     |      |      |  |
|---------|-----|-----|-----|-----|--|-----|------|------|--|
| SPBLND1 | 130 | 110 | 120 | CUB |  | 227 | 4.05 | 4.05 |  |
|         | 1.0 | 0.0 | 0.0 | 110 |  |     |      |      |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of blended spline. (Integer > 0)   |
| SID1       | Identification number of first spline (may be a blended spline). (Integer > 0)   |
| SID2       | Identification number of second spline (may be a blended spline). (Integer > 0)  |
| OPT        | Blending Option:<br>WAVG      Weighted Average (Default)<br>LIN      Linear Blending Functions<br>CUB      Cubic Blending Functions                              |
| W1         | Weight to be used with first spline. ( $0.0 < \text{Real} \leq 1.0$ ; Default = 0.5) (Used only with option WAVG)  |
| GID        | Identification number of an aerodynamic grid to be used as reference grid. (Integer > 0)   |
| D1         | Blending Depth of first spline. (Real > 0.0)   |
| D2         | Blending Depth of second spline. (Real > 0.0)  |
| X1, X2, X3 | Components of a direction vector v, in the coordinate system defined by CID, which is used to measure the distance from the reference grid. (See Remark 3.)      |
| CID        | Identification number of a rectangular coordinate system used to define the direction vector. (Integer > 0; Default = 0; indicating the basic coordinate system) |

**Remarks:**

1. The blending depth defines the maximum distance from the reference grid point of an aerodynamic grid point to be used in blending. It is also needed to evaluate the blending functions.
2. With option WAVG, the definition of GID, D1, D2 and the direction vector is optional. The weight W2 to be used with the second spline is computed from  $W2 = 1 - W1$ .

- The blended displacement is computed from

$$u_b = f_1(x)u_1 + f_2(x)u_2$$

where  $f_1(x)$  and  $f_2(x)$  are the blending functions (see [Figure 9-157](#)) and  $x$  is the distance from the reference grid point, measured in the direction of the direction vector  $v$ . Functions  $f_1(x)$  and  $f_2(x)$  sum up to 1.

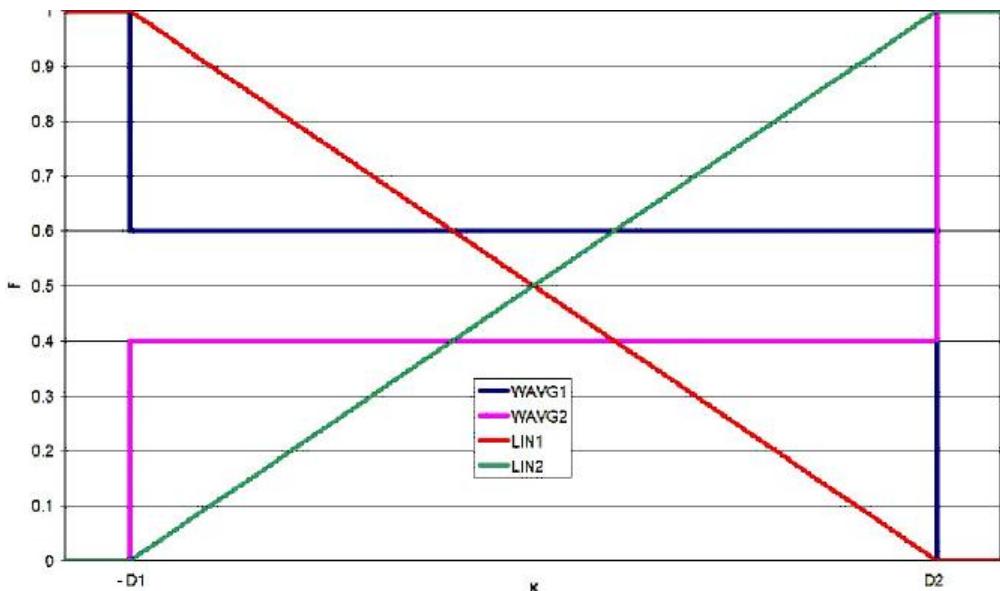


Figure 9-157 Blending Functions

- If the overlap region extends beyond  $-D1 < x < D2$ , then
 
$$f_1(x) = 1.0 \text{ and } f_2(x) = 0 \text{ for } x < -D1$$

$$f_1(x) = 0 \text{ and } f_2(x) = 1.0 \text{ for } x > D2$$
- The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
- If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

**SPBLND2**

## Curve Based Spline Blending

Defines a curve based blending of two splines.

**Format:**

| 1       | 2   | 3    | 4    | 5   | 6      | 7  | 8  | 9 | 10 |
|---------|-----|------|------|-----|--------|----|----|---|----|
| SPBLND2 | SID | SID1 | SID2 | OPT | AELIST | D1 | D2 |   |    |

**Example:**

|         |     |     |     |     |   |     |     |  |  |
|---------|-----|-----|-----|-----|---|-----|-----|--|--|
| SPBLND2 | 130 | 110 | 120 | LIN | 4 | 1.5 | 2.5 |  |  |
|---------|-----|-----|-----|-----|---|-----|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Identification number of blended spline. (Integer > 0)  |
| SID1       | Identification number of first spline (may be a blended spline). (Integer > 0)  |
| SID2       | Identification number of second spline (may be a blended spline). (Integer > 0)   |
| OPT        | Blending Option:<br>LIN Linear Blending Functions (Default)<br>CUB Cubic Blending Functions                               |
| AELIST     | Identification number of an AELIST entry listing the aerodynamic grid points that define a reference curve. (Integer > 0) |
| D1         | Blending Depth of first spline. (Real > 0.0)  |
| D2         | Blending Depth of second spline. (Real > 0.0)   |

**Remarks:**

1. The blending depth defines the maximum value of the distance of an aerodynamic grid point from the reference curve. It is also needed to evaluate the blending functions.
2. Blending functions are evaluated based on the distance of an aerodynamic grid point from the reference curve.
3. The reference curve is approximated by a polygon through the grid points listed on the AELIST entry referenced by AELIST. The list may contain coincident grid points. The order of the grid points is arbitrary.
4. The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
5. If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

**SPC****Single-Point Constraint**

Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).

**Format:**

| 1   | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-----|-----|----|----|----|----|----|----|---|----|
| SPC | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

**Example:**

|     |   |    |   |      |   |  |  |  |  |
|-----|---|----|---|------|---|--|--|--|--|
| SPC | 2 | 32 | 3 | -2.6 | 5 |  |  |  |  |
|-----|---|----|---|------|---|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of the single-point constraint set. (Integer > 0)  |
| Gi         | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. See Remark 8. ( $0 \leq$ Integer $\leq 6$ ; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 or 1 applies to scalar points and 1 through 6 applies to grid points.) |
| Di         | Value of enforced motion for components Gi at grid Ci. (Real; Default = 0.0)   |

**Remarks:**

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
3. Single-point forces of constraint are recovered during stress data recovery.
4. From 1 to 12 degrees-of-freedom may be specified on a single entry.
5. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
6. For reasons of efficiency, the SPCD entry is the preferred method for applying enforced motion rather than the Di field described here.
7. For SOL 400, the SPC entry requests enforced total displacement (Di) while the SPC1 entry requests null enforced relative displacements for a step. See the SPCD and SPCR entries for additional information.

| <b>Enforced Relative Displacement</b> |   |
|---------------------------------------|---|
| SPCR*                                 | associated with SPC1 or GRID/PS                                       |
| <b>Enforced Total Displacement</b>    |   |
| SPCD                                  | associated with SPC and overwrite the value of enforced motion on SPC |
| SPCD                                  | associated with SPC1 or GRID/PS                                       |
| SPC                                   | No association required   |

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPC entry requests enforced total displacement just like SPCD, however SPC may only be used in SOL 600 to enforce displacements if there is just one subcase in the analysis. If two or more subcases exist, SPCD or SPCR must be used to prescribe the enforced motion.

8. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
9. In thermal analysis, this entry specifies a constant temperature boundary condition applied on the selected grid or scalar point. For SOL 400, transient thermal analysis with any time-varying boundary condition, a constant temperature condition should be specified using the SPCD and SPC1 Bulk Data entries. For SOLs 153 and 159 thermal analysis, no TEMPBC Bulk Data entries with TYPE="STAT" may be specified with this entry.

**SPC1**

## Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

**Format:**

| 1    | 2   | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|------|-----|----|----|--------|----|----|----|----|----|
| SPC1 | SID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|      | G7  | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|      |   |   |   |   |    |   |   |   |  |
|------|---|---|---|---|----|---|---|---|--|
| SPC1 | 3 | 2 | 1 | 3 | 10 | 9 | 6 | 5 |  |
|      | 2 | 8 |   |   |    |   |   |   |  |

**Alternate Format and Example:**

|      |     |       |    |        |    |  |  |  |  |
|------|-----|-------|----|--------|----|--|--|--|--|
| SPC1 | SID | C     | G1 | "THRU" | G2 |  |  |  |  |
| SPC1 | 313 | 12456 | 6  | THRU   | 32 |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Identification number of single-point constraint set. (Integer > 0)   |
| C          | Component numbers. See Remark 7. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0, 1 or blank for scalar points.) |
| Gi         | Grid or scalar point identification numbers. (Integer > 0 or "THRU"; For "THRU" option, G1 < G2.)   |

**Remarks:**

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Enforced displacements are available via this entry when used with the recommended SPCD entry.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
5. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
6. For SOL 400, the SPC1 entry requests null enforced relative displacement for a step while the SPC entry requests enforced total displacements. SPC1 can always hold the displacement from the previous STEP. See the SPC, SPCD and SPCR entries for additional information.

| Enforced Relative Displacement |                         |                 |
|--------------------------------|-------------------------|-----------------|
| SPCR*                          | associated              | SPC1 or GRID/PS |
| Enforced Total Displacement    |                         |                 |
| SPCD                           | associated              | SPC             |
| SPCD                           | associated              | SPC1 or GRID/PS |
| SPC                            | No association required |                 |

In SOL 400 if a “control” GRID has an applied load associated with it in the previous STEP and if the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is still to use the SPCR, however, the user may apply instead an SPC1 containing the “control” grid ID.

7. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
8. In thermal analysis, this entry is used with SPCD to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPC Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE=“TRAN” for SOL 159.

**SPCADD****Single-Point Constraint Set Combination**

Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.

**Format:**

| 1      | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|-----|----|--------|----|----|----|----|----|----|
| SPCADD | SID | S1 | S2     | S3 | S4 | S5 | S6 | S7 |    |
|        | S8  | S9 | -etc.- |    |    |    |    |    |    |

**Example:**

|        |     |   |   |   |   |  |  |  |  |
|--------|-----|---|---|---|---|--|--|--|--|
| SPCADD | 101 | 3 | 2 | 9 | 1 |  |  |  |  |
|--------|-----|---|---|---|---|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Single-point constraint set identification number. (Integer > 0)   |
| Si         | Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries. (Integer > 0) |

**Remarks:**

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.
3. The Si values must be unique.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.
5. SPCADD entries take precedence over SPC entries. If both have the same SID, only the SPCADD entry will be used.

**SPCAX**

## Conical Shell Single-Point Constraint

Defines a set of single-point constraints or enforced displacements for conical shell coordinates.

**Format:**

| 1     | 2   | 3   | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|-----|---|---|---|---|---|----|
| SPCAX | SID | RID | HID | C | D |   |   |   |    |

**Example:**

|       |   |   |   |    |     |  |  |  |  |
|-------|---|---|---|----|-----|--|--|--|--|
| SPCAX | 2 | 3 | 4 | 13 | 6.0 |  |  |  |  |
|-------|---|---|---|----|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of a single-point constraint set. (Integer $> 0$ )               |
| RID        | Ring identification number. See RINGAX entry. (Integer $\geq 0$ )                      |
| HID        | Harmonic identification number. (Integer $\geq 0$ )                                    |
| C          | Component identification number. (Any unique combination of the Integers 1 through 6.) |
| D          | Enforced displacement value. (Real)  |

**Remarks:**

1. SPCAX is allowed only if an AXIC entry is also present.
2. Single-point constraint sets must be selected with the Case Control command SPC = SID.
3. Coordinates appearing on SPCAX entries may not appear on MPCAX, SUPAX, or OMITAX entries.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

**SPCD****Enforced Motion Value**

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

**Format:**

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| SPCD | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

**Example:**

|      |     |    |   |      |   |  |     |  |  |
|------|-----|----|---|------|---|--|-----|--|--|
| SPCD | 100 | 32 | 3 | -2.6 | 5 |  | 2.9 |  |  |
|------|-----|----|---|------|---|--|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number of the SPCD entry. (Integer > 0)   |
| Gi         | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component numbers. See Remark 10. ( $0 \leq$ Integer $\leq 6$ ; any unique combination of Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points) |
| Di         | Value of enforced motion for components Ci at grid Gi. (Real)  |

**Remarks:**

1. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. For SOL 600, it may also be selected by Li on the Bulk Data entry LOAD.
2. In dynamic analysis, the selection of SID is determined by the presence of the LOADSET request in Case Control as follows:
  - There is no LOADSET request in Case Control  
SID is selected by the EXCITEID field of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field
  - There is a LOADSET request in Case Control  
SID is selected by LID in the selected LSEQ entries that correspond to the EXCITEID entry of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field.
3. The SPCD entry is not supported in SOL 200.
4. A global coordinate (Gi and Ci) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command. This requirement is optional for SOL 600.

5. Enforced motion may be specified on both SPCD and SPC Bulk Data entries. The SPCD entry is selected according to Remark 2 and the SPC entry is selected by the SPC Case Control command. The Di values from both of the selected SPCD and SPC entries will be combined to form a static (or spatial) loading condition. But if a degree-of-freedom is duplicated on both SPCD and SPC then the value of Di on the selected SPCD entry will override the value of Dion the selected SPC entry. In addition, any degrees-of-freedom from the selected SPC entries and not duplicated will still be included in the loading condition. This is illustrated in the examples below.

- Static analysis: In the example below in SUBCASE 11, D=1.0 at GRID 101 overrides D=1.2 on the SPC entry. But GRID 201 is not specified on the selected SPCD and therefore D=1.2 from the SPC entry is also enforced. Similary, in subcase 12, D=1.0 is enforced at GRID 201 and D=1.2 at GRID 101.

```
SPC=77
SUBCASE 11
LOAD=701
SUBCASE 12
LOAD=702
BEGIN BULK
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2
```

- Dynamic analysis: In the example below, there are two unique EXCITEIDs specified on TLOAD2 and thusly two static loading conditions are created and identical to the conditions created in the static analysis example above.

```
SPC=77
DLOAD=201
BEGIN BULK
DLOAD,201,1.0,1.0,401,1.0,402
TLOAD2, 401, 701, , DISP, 0., 10.0, 60.,90.
TLOAD2, 402, 702, , DISP, 0., 10.0, 20.,270.
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2
```

6. The LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600.
7. In static analysis, this method of applying enforced displacements is more efficient than the SPC entry when more than one enforced displacement condition is applied. It provides equivalent answers.

8. In dynamic analysis, this direct method of specifying enforced motion is more accurate, efficient and elegant than the large mass and Lagrange multiplier techniques.
9. For SOL 400, the SPCD entry requests enforced total displacement for a STEP while the SPCR entry requests enforced relative displacements. SPCD is the position at the end of the applied STEP for ANALYSIS=NLSSTAT.

| <b>Enforced Relative Displacement</b> |   |
|---------------------------------------|---|
| SPCR*                                 | associated with SPC1 or GRID/PS                                       |
| <b>Enforced Total Displacement</b>    |   |
| SPCD                                  | associated with SPC and overwrite the value of enforced motion on SPC |
| SPCD                                  | associated with SPC1 or GRID/PS                                       |
| SPC                                   | No association required   |

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPCD entry requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR requests displacement or rotation for the current subcase relative to that of the previous subcase. SPCD may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.

10. For SOL 400 using heat shell element with linear or quadratic nodal distribution, see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID).
11. In thermal analysis, this entry is used with SPC1 to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPCD Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE="TRAN" for SOL 159 and SOL 600.
12. If it is desired to enforce motion on the boundary point (a-set) of a superelement then define a high stiffness element attached to the boundary point and apply the SPCD to the opposite point of the high stiffness element. High stiffness elements may be easily defined with CBUSH or CBAR.

**SPCOFF**

Excludes Degrees-of-Freedom from the AUTOSPC Operation

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

**Format:**

| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|----|----|----|----|----|----|----|----|----|
| SPCOFF | G1 | C1 | G2 | C2 | G3 | C3 | G4 | C4 |    |

**Example:**

|        |    |     |   |   |  |  |  |  |  |
|--------|----|-----|---|---|--|--|--|--|--|
| SPCOFF | 32 | 436 | 5 | 1 |  |  |  |  |  |
|--------|----|-----|---|---|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| Gi         | Grid or scalar point identification number. (Integer > 0)   |
| Ci         | Component numbers. (Integer 0, 1 or blank for scalar points; Integers 1 through 6 with no embedded blanks for grid points.) |

**Remarks:**

- Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
- Degrees-of-freedom that are specified as both SPC and SPCOFF will be considered as SPC.

**SPCOFF1**

Excludes DOF's from AUTOSPC Processing, Alternate Form

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

**Format:**

| 1       | 2  | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|----|--------|----|----|----|----|----|----|
| SPCOFF1 | C  | G1 | G2     | G3 | G4 | G5 | G6 | G7 |    |
|         | G8 | G9 | -etc.- |    |    |    |    |    |    |

**Example:**

|         |   |   |   |    |   |   |   |   |  |
|---------|---|---|---|----|---|---|---|---|--|
| SPCOFF1 | 2 | 1 | 3 | 10 | 9 | 6 | 5 | 4 |  |
|         | 8 |   |   |    |   |   |   |   |  |

**Alternate Format and Example:**

|         |       |    |        |    |  |  |  |  |  |
|---------|-------|----|--------|----|--|--|--|--|--|
| SPCOFF1 | C     | G1 | "THRU" | G2 |  |  |  |  |  |
| SPCOFF1 | 12456 | 6  | THRU   | 32 |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| C          | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points.) |
| Gi         | Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for "THRU" option, G1 < G2.)   |

**Remarks:**

- Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
- Degrees-of-freedom that are both specified as SPC and SPCOFF will be considered as SPC.
- If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.

**SPCR****Enforced Relative Motion Value**

Defines an enforced relative displacement value for a load step in SOL 400 and SOL 600.

**Format:**

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| SPCR | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

**Example:**

|      |     |    |     |      |   |  |     |  |  |
|------|-----|----|-----|------|---|--|-----|--|--|
| SPCR | 100 | 32 | 436 | -2.6 | 5 |  | 2.9 |  |  |
|------|-----|----|-----|------|---|--|-----|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number of the SPCR entry. (Integer > 0)   |
| Gi         | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. ( $0 \leq \text{Integer} \leq 6$ ; up to six unique Integers may be placed in the field with no embedded blanks, a blank or 0 is treated the same as 1.) |
| Di         | Value of enforced motion for Gi and Ci. (Real)   |

**Remarks:**

1. SPCR requests relative motion for a load step with respect to the previous step in SOL 400 or subcase in SOL 600. SPCR is the companion entry to SPCD. SPCD requests total motion while SPCR requests relative motion. SPCR is the position at activation of the boundary condition.

| Enforced Relative Displacement |   |
|--------------------------------|---|
| SPCR*                          | associated with SPC1 or GRID/PS                                       |
| Enforced Total Displacement    |   |
| SPCD                           | associated with SPC and overwrite the value of enforced motion on SPC |

| Enforced Relative Displacement |                                 |
|--------------------------------|---------------------------------|
| SPCD                           | associated with SPC1 or GRID/PS |
| SPC                            | No association required         |

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPCR entry requests displacement or rotation for the current subcase relative to that of the previous subcase. The SPCD requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.

2. The enforced motion for a step, for SOL 400 or subcase for SOL 600, can be either total value or relative value. For SOL 400, SPC and SPCD request total enforced motion. SPC1 and SPCR request the relative value. For example, if a DOF is specified on a SPCR with 0.0 for step 2, the relative displacement of this DOF for step 2 with respect to step 1 is 0.0. The total displacement of step 2 is 0.2 if the solution of step 1 for this DOF is 0.2.
3. The SCPD and SPCR entries can have the same SID, but they cannot be specified on the same DOF. A user fatal error will be issued if SPCD and SPCR are specified on the same DOF.
4. In the static solution sequences, the SID of the SPCR entry (SID), same as SPCD, is selected by the LOAD Case Control command.
5. For SOL 400, a global coordinate (Gi and CI) referenced on this entry must also be referenced on a SPC1 Bulk Data entry and selected by the SPC Case Control command. Please note that, for this purpose, SPC cannot be used together with SPCR. If SPC is used, a user fatal error will be issued.
6. For SOL 400, the LOAD Bulk Data entry will not combine an SPCR load entry.
7. For SOL 600 if SPCR is entered, PARAM,MARCTOTT,1 must also be entered.

**SPHERE**

Defines the Shape of a Sphere

Spherical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3 | 4 | 5 | 6 | 7      | 8 | 9 | 10 |
|--------|-----|---|---|---|---|--------|---|---|----|
| SPHERE | VID |   | X | Y | Z | RADIUS |   |   |    |

**Example:**

|        |     |  |    |    |    |    |  |  |  |
|--------|-----|--|----|----|----|----|--|--|--|
| SPHERE | 100 |  | 1. | 1. | 1. | .5 |  |  |  |
|--------|-----|--|----|----|----|----|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| VID        | Number of the sphere. (Integer > 0; Required)                  |
| X, Y, Z    | Coordinates of the center of the sphere. (Real; Default = 0.0) |
| RADIUS     | Radius of the sphere. (Real > 0; Required)                     |

**SPLINE1**

## Surface Spline Methods

Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

**Format:**

| 1       | 2     | 3     | 4    | 5    | 6    | 7  | 8    | 9     | 10 |
|---------|-------|-------|------|------|------|----|------|-------|----|
| SPLINE1 | EID   | CAERO | BOX1 | BOX2 | SETG | DZ | METH | USAGE |    |
|         | NELEM | MELEM |      |      |      |    |      |       |    |

**Example:**

|         |   |     |     |     |    |    |  |  |  |
|---------|---|-----|-----|-----|----|----|--|--|--|
| SPLINE1 | 3 | 111 | 115 | 122 | 14 | 0. |  |  |  |
|---------|---|-----|-----|-----|----|----|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique spline identification number. (Integer > 0)  |
| CAERO      | Aero-element (CAEROi entry ID) that defines the plane of the spline. (Integer > 0)  |
| BOX1, BOX2 | First and last box with motions that are interpolated using this spline; see Remark 3. when using Mach Box method. (Integer > 0; BOX2 ≥ BOX1)   |
| SETG       | Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)   |
| DZ         | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| METH       | Method for the spline fit. IPS, TPS or FPS. See Remark 1. (Character; Default = IPS)  |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 5. (Character; Default = BOTH) |
| NELEM      | The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default = 10)  |
| MELEM      | The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default = 10)  |

**Remarks:**

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.
2. The interpolated points ( $k$ -set) will be defined by aero boxes. Figure 9-158 shows the cells for which  $u_k$  is interpolated if BOX1 = 115 and BOX2 = 122.

|     |     |     |     |
|-----|-----|-----|-----|
| 111 | 114 | 117 | 120 |
| 112 | 115 | 118 | 121 |
| 113 | 116 | 119 | 122 |

Figure 9-158 SPLINE1 Entry Example

3. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
4. When using the Mach Box method, BOX1 and BOX2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. BOX1 and BOX2 do not refer to Mach Boxes.
5. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$F_g = [GP_{kg}]^T \{P_k\}$  (FORCE/BOTH splines are in the transform)

$U_k = [GD_{kg}] \{U_g\}$  (DISP/BOTH splines are in the transform)

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship  $[GP_{gk}]^T = [GD_{kg}]$  satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCES from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). MSC Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

6. The SPLINE1 EID must be unique with respect to all SPLINEi entries.

**SPLINE2**

## Linear Spline

Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

**Format:**

| 1       | 2    | 3     | 4   | 5     | 6    | 7  | 8    | 9   | 10 |
|---------|------|-------|-----|-------|------|----|------|-----|----|
| SPLINE2 | EID  | CAERO | ID1 | ID2   | SETG | DZ | DTOR | CID |    |
|         | DTHX | DTHY  |     | USAGE |      |    |      |     |    |

**Example:**

|         |    |   |    |    |    |    |     |   |  |
|---------|----|---|----|----|----|----|-----|---|--|
| SPLINE2 | 5  | 8 | 12 | 24 | 60 | 0. | 1.0 | 3 |  |
|         | 1. |   |    |    |    |    |     |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique spline identification number. (Integer > 0)  |
| CAERO      | Aero panel or body (CAERO <i>i</i> entry ID) that is to be interpolated. (Integer > 0)  |
| ID1, ID2   | First and last box or body element whose motions are interpolated using this spline. See Remark 6. when using the Mach Box method. (Integer > 0; ID2 ≥ ID1)   |
| SETG       | Refers to an SET <i>i</i> entry that lists the structural grid points to which the spline is attached. (Integer > 0)  |
| DZ         | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| DTOR       | Torsional flexibility ratio ( $EI/GJ$ ). (Real $> 0.0$ ; Default = 1.0; use 1.0 for bodies.)  |
| CID        | Rectangular coordinate system for which the y-axis defines the axis of the spline. Not used for bodies, CAERO2. (Integer $\geq 0$ )   |
| DTHX, DTHY | Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (in-plane bending rotations); however, it is not used for bodies. DTHY is for rotation about the spline's y-axis (torsion); however, it is used for slope of bodies. (Real) |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 9. (Character; Default = BOTH)   |

**Remarks:**

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.

3. The flexibilities DZ, DTHX, and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment). See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE2 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, ID1 and ID2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. ID1 and ID2 do not refer to Mach Boxes.
7. DTOR is the ratio of rotational to linear deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur.
8. If a SPLINE2 element only references one grid point, the job will fail without a message in the GI Module.
9. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

**SPLINE3****Aeroelastic Constraint Equation**

Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

**Format:**

| 1       | 2   | 3     | 4     | 5    | 6     | 7  | 8  | 9     | 10 |
|---------|-----|-------|-------|------|-------|----|----|-------|----|
| SPLINE3 | EID | CAERO | BOXID | COMP | G1    | C1 | A1 | USAGE |    |
|         | G2  | C2    | A2    |      | -etc. |    |    |       |    |

**Example:**

|         |      |     |      |   |   |   |     |  |  |
|---------|------|-----|------|---|---|---|-----|--|--|
| SPLINE3 | 7000 | 107 | 109  | 6 | 5 | 3 | 1.0 |  |  |
|         | 43   | 5   | -1.0 |   |   |   |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| EID        | Unique spline identification number. (Integer > 0)   |
| CAERO      | Identification number of the macro-element on which the element to be interpolated lies. (Integer > 0)   |
| BOXID      | Identification number of the aerodynamic element; i.e., the box number. (Integer > 0)  |
| COMP       | The component of motion to be interpolated. See Remark 4. (One of the Integers 1, 2, 3, 4, 5, or 6.)   |
| Gi         | Grid point identification number of the independent grid point. (Integer > 0)  |
| Ci         | Component numbers in the displacement coordinate system. (One of the Integers 1 through 6 for grid points, or 0 for scalar points.)  |
| Ai         | Coefficient of the constraint relationship. (Real)   |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 6. (Character; Default = BOTH). |

**Remarks:**

1. The independent grid points and components must refer to degrees-of-freedom in the g-set.
2. The constraint is given by

$$u_d = \sum A_i u_i$$

where:

$u_d$  = value of the dependent component of the aerodynamic element

$u_i$  = displacement at grid Gi, component Ci.

3. The SPLINE3 EID must be unique with respect to all SPLINEi entries.

4. The allowable components by CAEROi entry type are indicated by an “X” in the table below:

| Entry Type  | COMP |   |   |   |   |
|-------------|------|---|---|---|---|
|             | 1    | 2 | 3 | 5 | 6 |
| CAERO1      |      |   | X | X |   |
| CAERO2      |      | X | X | X | X |
| CAERO3      |      |   | X |   |   |
| CAERO4      |      |   | X | X | X |
| CAERO5      |      |   | X | X | X |
| 3D Geometry | X    | X | X | X | X |

COMP = 2: lateral displacement

COMP = 3 transverse displacement

COMP = 5: pitch angle

COMP = 6: relative control angle for CAERO4 and CAERO5 yaw angle for CAERO2.

For general 3D aerodynamic geometries the components numbers refer to axes of the Aerodynamic Coordinate System ( $u_x, u_y, u_z, \theta_x, \theta_y, \theta_z$ ).

5. For Strip theory and Piston theory, the COMP = 6 control surface relative angle is positive when the trailing edge has a negative z-deflection in the element coordinate system (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).
6. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

**SPLINE4**

## Surface Spline Methods

Defines a curved surface spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

**Format:**

| 1       | 2     | 3     | 4      | 5     | 6    | 7  | 8    | 9     | 10 |
|---------|-------|-------|--------|-------|------|----|------|-------|----|
| SPLINE4 | EID   | CAERO | AELIST |       | SETG | DZ | METH | USAGE |    |
|         | NELEM | MELEM | FTYPE  | RCORE |      |    |      |       |    |

**Example:**

|         |   |     |     |  |    |    |     |  |  |
|---------|---|-----|-----|--|----|----|-----|--|--|
| SPLINE4 | 3 | 111 | 115 |  | 14 | 0. | IPS |  |  |
|---------|---|-----|-----|--|----|----|-----|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Unique spline identification number. (Integer > 0)  |
| CAERO      | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)  |
| AELIST     | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)  |
| SETG       | Identification number of a SET1 entry that lists the structural grid points to which the spline is attached. (Integer > 0)  |
| DZ         | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| METH       | Spline method: <ul style="list-style-type: none"> <li>IPS      Infinite Plate Spline (Default)</li> <li>TPS      Thin Plate Spline</li> <li>FPS      Finite Plate Spline</li> <li>RIS      Radial Interpolation Spline</li> </ul> |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. See Remark 3. (Character, Default = BOTH)                      |
| NELEM      | The number of FE elements along the local x-axis if using the FPS option. (Integer > 0; Default = 10)   |
| MELEM      | The number of FE elements along the local y-axis if using the FPS option. (Integer > 0; Default = 10)   |
| FTYPE      | Selects the radial interpolation function to be used with the RIS option: <ul style="list-style-type: none"> <li>WF0      C0 continuous Wendland function</li> </ul>  |

| Descriptor | Meaning  |
|------------|--|
| WF2        | C2 continuous Wendland function (Default)                                    |
| RCORE      | Radius of support of radial interpolation function. (Real > 0.0; no Default) |

**Remarks:**

1. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AFFACT entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. NELEM and MELEM are used only for the METH=FPS and are required only when the defaults are not adequate.
5. FTTYPE and RCORE used only with METH=RIS. FTTYPE=WF0 uses a Wendland function:

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^2$$

while FTTYPE = WF2 uses

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^4 \left(4\frac{r}{r_c} + 1\right)$$

where

$$(y)_t = \begin{cases} y & \text{if } y > 0 \\ o & \text{if } y < 0 \end{cases}$$

**SPLINES5**

## Linear Spline

Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries.

**Format:**

| 1        | 2    | 3     | 4      | 5     | 6    | 7  | 8     | 9     | 10 |
|----------|------|-------|--------|-------|------|----|-------|-------|----|
| SPLINES5 | SID  | CAERO | AELIST |       | SETG | DZ | DTOR  | CID   |    |
|          | DTHX | DTHY  |        | USAGE | METH |    | FTYPE | RCORE |    |

**Example:**

|          |    |   |    |      |    |  |  |   |  |
|----------|----|---|----|------|----|--|--|---|--|
| SPLINES5 | 5  | 8 | 12 |      | 60 |  |  | 3 |  |
|          | 1. |   |    | BOTH |    |  |  |   |  |

| Descriptor    | Meaning  |
|---------------|--|
| SID           | Unique spline identification number (Integer > 0)  |
| CAERO         | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)   |
| AELIST        | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 6. (Integer > 0)   |
| SETG          | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)   |
| DZ            | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)   |
| DTOR          | Torsional flexibility ratio (EI/GJ) for the bending in the zy-plane. This value is ignored for slender bodies since they have no torsion; see Remark 7. (Real > 0.0; Default = 1.0; ignored for CAERO2 bodies.)        |
| CID           | Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer $\geq 0$ )  |
| DTHX,<br>DTHY | Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (the bending rotations). DTHY is for rotation about the spline's y-axis (torsion); however, it is used for bending of bodies. (Real) |
| USAGE         | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 8. (Character; Default = BOTH)                            |
| METH          | Spline method:<br><br>BEAM      Beam Spline (Default)<br>RIS      Radial Interpolation Spline  |
| FTYPE         | Selects the radial interpolation function to be used with the RIS option:  |

| Descriptor | Meaning  |
|------------|--|
| WF0        | C0 continuous Wendland function  |
| WF2        | C2 continuous Wendland function. (Default)                                   |
| RCORE      | Radius of support of radial interpolation function. (Real > 0.0; no Default) |

**Remarks:**

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the interpolating beam (y-axis) is parallel to the x-axis of the aerodynamic coordinate system; the z-axis is taken from the referenced CID and x is made orthogonal.
3. The flexibilities DZ, DTHX and DTHY are used for smoothing. (Zero attachment flexibility values imply rigid attachment; i.e., no smoothing, whereas negative values of DTHX or DTHY imply infinity, therefore, no attachment.) See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE5 EID must be unique with respect to all SPLINEi entries.
6. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
7. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
8. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
9. FTYPE and RCORE are only used for METHOD=RIS. See Remark 5. on the SPLINE 4 entry for descriptions of the Wendland functions.

**SPLINE6**

## 3D Finite Surface Spline

Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.

**Format:**

| 1       | 2      | 3      | 4      | 5      | 6      | 7   | 8       | 9     | 10 |
|---------|--------|--------|--------|--------|--------|-----|---------|-------|----|
| SPLINE6 | EID    | CAERO  | AELIST |        | SETG   | DZ  | METHOD  | USAGE |    |
|         | VSTYPE | VSLIST | I2VNUM | D2VNUM | METHVS | DZR | METHCON | NGRID |    |
|         | ELTOL  | NCYCLE | AUGWEI |        |        |     |         |       |    |

**Example:**

|         |      |   |    |   |     |  |      |      |  |
|---------|------|---|----|---|-----|--|------|------|--|
| SPLINE6 | 5    | 8 | 12 |   | 60  |  | FPS6 | DISP |  |
|         | AERO | 2 | 4  | 4 | VS6 |  |      |      |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique spline identification number. (Integer > 0)  |
| CAERO      | Aero panel (CAEROi entry ID) that is to be interpolated. See Remarks 2. and 4. (Integer > 0 or blank)   |
| AELIST     | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)  |
| SETG       | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)  |
| DZ         | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| METHOD     | Method for the spline fit. Either FPS3 or FPS6. See Remark 5. (Character; Default = FPS6)   |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 3. (Character; Default = BOTH)   |
| VSTYPE     | Virtual surface connectivity type. Either AERO or STRUC. (Character; Default = AERO)  |
| VSLIST     | Identification number of an AELIST entry listing quadrilateral and/or triangular shell elements of the VSTYPE mesh which define the connectivity of the virtual surface mesh. See Remark 2. (Integer > 0 or can be blank if CAERO is not blank and VSTYPE=AERO) |
| I2VNUM     | The minimum number of structural mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)  |
| D2VNUM     | The minimum number of aero mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)  |

| Descriptor | Meaning  |
|------------|--|
| METHVS     | Similar to METHOD, this field chooses whether or not to include the rotational degrees-of-freedom of virtual surface. Either VS6 or VS3. See Remark 5. (Character, Default VS6)                        |
| DZR        | Rotational attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)   |
| METHCON    | Method used to determine RBE3 connecting points between the meshes. Either NODEPROX or CIRCBIAS. See Remarks 7., 8. and 9. (Character; Default = NODEPROX)   |
| NGRID      | Number of closest grids that are used to determine the element list that is used to define the RBE3 elements. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Integer $> 0$ ; Default = 1)     |
| ELTOL      | Tolerance used to determine whether or not a node projects onto an element of the mesh. Specified as % of element size. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real; Default = 100.0) |
| NCYCLE     | Maximum number of cycles used to find elements onto which the nodes project. Only valid for METHCON=CIRCBIAS. See Remark 7. and 9. (Integer $> 0$ ; Default = 3)                                       |
| AUGWEI     | RBE3 weighting factor augmentation parameter. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real $\geq 0.0$ ; Default = 0.0)   |

**Remarks:**

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating surface and the connections to the dependent and independent grids.
2. If an aerodynamic mesh is entered using AEGRID/AETRIA3/AEQUAD4 entries, the CAERO ID is not required and the AELIST refers to the element ID's of the aero mesh. In this scenario, if VSTYPE=AERO, VSLIST must point to an AELIST that defines the virtual surface mesh by identifying the aerodynamic elements that make up the surface. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
5. The METHOD option provides a choice in using all 6 degrees of freedom (FPS6) on the independent points or only the translational degrees of freedom (FPS3) in connecting between the virtual surface and the independent points. Similarly, there is a choice in connecting the virtual surface to the dependent points (METHVS).

6. The connection between the independent points (structural) and the dependent points (aero) is made through a virtual surface whose mesh is defined by elements listed in the VSLIST (these are either AERO box id's or STRUCtural shell elements (CQUAD4, CQUADR, CTRIA3 ,CTRIAR)).
7. To bind the points to the virtual surface, a connection is made between the points and the surface using automatically generated virtual RBE3 elements. Two methods exist to choose which independent mesh points are connected to each dependent mesh point: nodal-proximity (NODEPROX) and circular bias (CIRCBIAS).
8. The nodal proximity method selects the closest independent mesh points to each dependent mesh point. The actual number of points will depend on the user inputs I2VNUM and D2VNUM as well as collinearity checks. Larger values will spread the connectivity (smearing). Smaller values allow for more concentration (with additional points added as necessary for collinearity).
9. The circular bias method uses elements of the virtual mesh in an attempt to select independent mesh points that encircle each dependent mesh point. This method will be restricted to the case where the virtual mesh is the target mesh.

This method will do the following:

- For each splined dependent mesh node, find the closest NGRID splined independent mesh node(s).
- Assemble the list of virtual mesh elements that use the closest node(s).
- Check each of these elements to see if the dependent node projects onto the element in the element's mean plane normal direction. Note that this check may be computationally expensive, so it is performed only to the "possible" elements, not the entire virtual mesh. The projection check will contain a user-defined tolerance, ELTOL, to expand the area of the element that is acceptable for a match.
- If the dependent node does not project onto any element, use the candidate element's nodes to expand the list of elements to check. Repeat the projection check (the original elements will not be rechecked). Repeat this process up to NCYCLE times.
- All elements that are found to encompass the dependent node (and there may be more than one due to curvature) will be selected to move forward.
- Assemble the list of all splined nodes that connect the selected elements.
- Generate RBE3 elements based on this node list. An optional user-defined input parameter, AUGWEI, will be used to augment the RBE3 weighting factors with the following formula:

$$\text{weight} = (\text{NE} - 1) \cdot \text{AUGWEI} + 1$$

where NE is the number of elements that are connected with the RBE3 node.

**SPLINE7**

## Finite Beam Spline

Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.

**Format:**

| 1       | 2   | 3     | 4      | 5     | 6      | 7   | 8    | 9     | 10 |
|---------|-----|-------|--------|-------|--------|-----|------|-------|----|
| SPLINE7 | EID | CAERO | AELIST |       | SETG   | DZ  | DTOR | CID   |    |
|         |     |       |        | USAGE | METHOD | DZR | IA2  | EPSBM |    |

**Example:**

|         |   |   |    |      |      |  |  |   |  |
|---------|---|---|----|------|------|--|--|---|--|
| SPLINE7 | 5 | 8 | 12 |      | 60   |  |  | 3 |  |
|         |   |   |    | BOTH | FBS6 |  |  |   |  |

| Descriptor | Meaning   |
|------------|---|
| EID        | Unique spline identification number. (Integer > 0)  |
| CAERO      | Aero panel (CAEROi entry ID) that is to be interpolated. See Remark 6. (Integer > 0 or blank)   |
| AELIST     | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)  |
| SETG       | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)  |
| DZ         | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| DTOR       | Ratio of the beam bending stiffness to the beam torsional stiffness. See Remark 3.. (Real > 0.0; Default = 1.0)   |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4. (Character, Default = BOTH) |
| CID        | Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz- planes for bending. Not used for bodies, CAERO2. (Integer $\geq 0$ )                                |
| METHOD     | Method for the spline fit. Either FBS3 or FBS6. See Remark 5. (Character; Default = FBS6)   |
| DZR        | Rotational attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)  |
| IA2        | Ratio of the beam bending stiffness to the beam extensional stiffness. (Real > 0.0; Default = 1.0)  |
| EPSBM      | Ratio of the minimum beam length to the total beam length. See Remark 7. (Real > 0.0; Default = 0.01)   |

**Remarks:**

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating beam and the connections to the dependent and independent grids.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.
3. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily translational deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
5. The FBS3 method will map only the three translational degrees of freedom. The FBS6 method will map all six degrees of freedom.
6. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
7. EPSBM is the minimum length of a SPLINE7 beam as a fraction of the total SPLINE7 FEM length. Thus, an EPSBM value of 0.01 (Default), would produce a minimum beam length that is 1% of the total FEM length. Defining a value of EPSBM that is smaller than 0.01 will decrease the minimum length and potentially increase the number of beams used to define the SPLINE7 FEM.

**SPLINEX****Externally-Evaluated Spline**

Defines the input for a spline that will be evaluated with a user-supplied procedure.

**Format:**

| 1       | 2      | 3      | 4       | 5      | 6      | 7      | 8 | 9     | 10 |
|---------|--------|--------|---------|--------|--------|--------|---|-------|----|
| SPLINEX | EID    | GROUP  | DGCOMP  | IGCOMP | DECOMP | IECOMP |   | USAGE |    |
|         | AELIST | AEFACT | AELISTC |        |        |        |   |       |    |

**Example:**

|         |     |          |        |        |  |  |  |      |  |
|---------|-----|----------|--------|--------|--|--|--|------|--|
| SPLINEX | 3   | SPLNGRP4 | GWNG1A | GWNG1S |  |  |  | BOTH |  |
|         | 101 | 201      | 301    |        |  |  |  |      |  |

**Descriptor****Meaning**

|         |   |
|---------|---|
| EID     | Element identification number. (Integer > 0)  |
| GROUP   | Group name to which the external spline type belongs. (Character; no Default)   |
| DGCOMP  | The name of an AECOMP or AECOMPL entry that defines the set of points for the dependent mesh. See Remarks 3. and 4. (Character; Default = Blank).   |
| IGCOMP  | The name of an AECOMP or AECOMPL entry that defines the set of points for the independent mesh. See Remarks 3. and 4. (Character; Default = Blank).   |
| DECOMP  | The name of an AECOMP or AECOMPL entry that defines the set of elements for the dependent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).                                     |
| IECOMP  | The name of an AECOMP or AECOMPL entry that defines the set of elements for the independent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).                                   |
| USAGE   | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 2. (Character; Default = BOTH) |
| AELIST  | ID of an AELIST that contains a list of user-defined integer data. See Remark 6. (Integer; no Default).   |
| AEFACT  | ID of an AEFACT that contains a list of user-defined real data. See Remark 6. (Integer; no Default).  |
| AELISTC | ID of an AELISTC that contains a list of user-defined character data. See Remark 6. (Integer; no Default).  |

**Remarks:**

1. The SPLINEX EID must be unique with respect to all SPLINEi entries.

2. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$F_g = [GPkg]^T \{Pk\}$  (FORCE/BOTH splines are in the transform)

$U_k = [GDkg] \{Ug\}$  (DISP/BOTH splines are in the transform)

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship  $[GPkg]^T = [GDkg]$  satisfied. The default behavior (BOTH for all splines) is compatible with version of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

3. Typically, for aero-to-structure splines "dependent" means "aerodynamic", and "independent" means "structural".
4. If the component defines a structural mesh, then the grid component may be left blank and the list of grids will be obtained from the element component member's connectivity. Both may not be left blank.
5. Structural elements referenced by DECOMP and IECOMP are limited to the following element types: CQUAD4, CQUADR, CTRIA3, CTRIAR. In one list, elements from the different types may not share the same ID.
6. The data that are defined on the AELIST, AEFACT, and AELISTC have no meaning to Nastran. These lists are generic containers for data that has meaning to the spline server. Note that the AELIST is limited to numbers greater than zero.

**SPLINRB****Rigid Body Spline**

Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

**Format:**

| 1       | 2   | 3     | 4      | 5     | 6  | 7  | 8  | 9  | 10 |
|---------|-----|-------|--------|-------|----|----|----|----|----|
| SPLINRB | SID | CAERO | AELIST | USAGE | G1 | C1 | G2 | C2 |    |
|         | G3  | C3    | G4     | C4    | G5 | C5 | G6 | C6 |    |

**Example:**

|         |     |    |   |      |      |        |  |  |  |
|---------|-----|----|---|------|------|--------|--|--|--|
| SPLINRB | 110 | 20 | 2 | BOTH | 1093 | 123456 |  |  |  |
|---------|-----|----|---|------|------|--------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique spline identification number. (Integer > 0)   |
| CAERO      | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0 or blank)  |
| AELIST     | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 3. (Integer > 0)   |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. (Character; Default = BOTH) |
| Gi         | Identification number of a structural grid point. (Integer > 0)  |
| Ci         | Component numbers: Any unique combination of the Integers 1 through 6 with no embedded blanks.   |

**Remarks:**

- Up to six structural grid points can be used to select exactly 6 structural degrees-of-freedom that define the motion of the rigid body.
- The selected degrees-of-freedom must define statically determinate supports of the rigid body.
- For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.

**SPOINT**

## Scalar Point Definition

Defines scalar points.

**Format:**

| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| SPOINT | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 | ID7 | ID8 |    |

**Example:**

|        |   |    |   |   |    |   |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|
| SPOINT | 3 | 18 | 1 | 4 | 16 | 2 |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|

**Alternate Format and Example:**

|        |     |        |     |  |  |  |  |  |  |
|--------|-----|--------|-----|--|--|--|--|--|--|
| SPOINT | ID1 | "THRU" | ID2 |  |  |  |  |  |  |
| SPOINT | 5   | THRU   | 649 |  |  |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| IDi        | Scalar point identification number. (0 < Integer < 100,000,000; For "THRU" option, ID1 < ID2) |

**Remarks:**

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASI, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.
2. All scalar point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points. However, duplicate scalar point identification numbers are allowed in the input.
3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.
4. If the alternate format is used, all scalar points ID1 through ID2 are defined.
5. For a discussion of scalar points, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) (p. 174) in the *MSC Nastran Reference Guide*.

**SPRELAX**

## Spline Relaxation

Defines relaxation of a spline based on an adjacent spline.

**Format:**

| 1       | 2    | 3    | 4     | 5    | 6     | 7 | 8 | 9 | 10 |
|---------|------|------|-------|------|-------|---|---|---|----|
| SPRELAX | SID1 | SID2 | LIST2 | DREF | LIST1 |   |   |   |    |

**Example:**

|         |     |     |    |     |  |  |  |  |  |
|---------|-----|-----|----|-----|--|--|--|--|--|
| SPRELAX | 140 | 130 | 50 | 5.0 |  |  |  |  |  |
|---------|-----|-----|----|-----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID1       | Identification number of spline to be modified (may be a blended spline and need not be unique). (Integer > 0)   |
| SID2       | Identification number of adjacent spline (may be a blended spline). (Integer > 0)  |
| LIST2      | Identification of an AELIST entry listing aerodynamic grid points that define the curve used in the relaxation . (Integer > 0) (See Remark 1.)               |
| DREF       | Reference Distance (Real > 0.) (See Remark 2.)   |
| LIST1      | Identification number of an AELIST entry listing aerodynamic grid points of the spline to be modified (optional). (Integer > 0; Default = 0) (See Remark 3.) |

**Remarks:**

1. The aerodynamic grid points referenced by LIST2 have to define a curve. The curve need not be contiguous, i.e., coincident grid points are allowed. The order of the grid points is arbitrary.
2. Displacements of spline 1 are modified according to

$$u_{1mod} = u_1 + f\left(\frac{r}{DREF}\right)(u_2 - u_1)$$

where  $r$  is the shortest distance of the aerodynamic grid point considered from the curve,  $u_2$  is the displacement from spline 2, interpolated to the position on the curve which is closest to the aerodynamic grid point considered, and function  $f$  is defined by

$$f(x) = \begin{cases} 1-x & \text{if } x \leq 1 \\ 0 & \text{if } x > 1 \end{cases}$$

3. If LIST1 is defined, only aerodynamic grid points contained in the referenced list are processed. Otherwise, all aerodynamic grid points of spline 1 are processed.

**STOCHAS**

## Randomization of Model Parameters

Specifies statistics used in randomization selected model parameters.

**Format:**

| 1       | 2   | 3      | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|-----|--------|-----|-----|---|---|---|---|----|
| STOCHAS | SID | PENTRY | CDF | CoV | m |   |   |   |    |
|         |     | MENTRY | CDF | CoV | m |   |   |   |    |
|         |     | CENTRY | CDF | CoV | m |   |   |   |    |
|         |     | LOADS  | CDF | CoV | m |   |   |   |    |
|         |     | SPCD   | CDF | CoV | m |   |   |   |    |

**Example 1: (Randomize all element and material properties with the default settings.)**

|         |     |        |  |  |  |  |  |  |  |
|---------|-----|--------|--|--|--|--|--|--|--|
| STOCHAS | 100 | PENTRY |  |  |  |  |  |  |  |
|         |     | MENTRY |  |  |  |  |  |  |  |

**Example 2: (Randomize material properties with CoV = 0.1 and loadings with CoV = 0.3 and default multipliers of standard deviations.)**

|         |     |        |       |     |  |  |  |  |  |
|---------|-----|--------|-------|-----|--|--|--|--|--|
| STOCHAS | 200 | LOADS  | GAUSS | 0.3 |  |  |  |  |  |
|         |     | MENTRY | GAUSS | 0.1 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique identification number that is selected by the STOCHASTICS Case Control command. (Integer > 0) |
| “PENTRY”   | Flag for randomizing real values on all the element property entries. (Character)                    |
| “MENTRY”   | Flag for randomizing real values on all the independent material property entries. (Character)       |
| “CENTRY”   | Flag for randomizing real values on all the connectivity entries. (Character)                        |
| “LOADs”    | Flag for randomizing real values on all the load entries. (Character)                                |
| “SPCD”     | Flag for randomizing real values on all the SPCD entries. (Character)                                |
| CDF        | Name of a cumulative distribution function. See Remark 2. (Character; Default = GAUSSIANS or blank). |
| CoV        | Coefficient of variance. (Real > 0; Default = 0.05)  |
| m          | Number of standard deviations. See Remark 3. (Real > 0; Default = 3.0)                               |

**Remarks:**

- At least one flag must exist but they can be placed in any order.
- Currently, only Gaussian distributions are supported.

3. The range of a random variable is defined as  $(\mu - m \cdot \sigma, \mu + m \cdot \sigma)$  where  $\mu$  is the mean of the random variable (or the value of one analysis model parameter on a Bulk Data entry),  $\sigma$  is the standard deviation that is related to  $\mu$ , CoV by  $\sigma = \text{CoV} \cdot \mu$  and  $m$  is the multiplier of the standard deviations.
4.  $m \cdot \text{CoV}$  must be  $< 1.0$ .

**SUPAX**

## Conical Shell Fictitious Support

Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.

**Format:**

| 1     | 2    | 3    | 4  | 5    | 6    | 7  | 8 | 9 | 10 |
|-------|------|------|----|------|------|----|---|---|----|
| SUPAX | RID1 | HID1 | C1 | RID2 | HID2 | C2 |   |   |    |

**Example:**

|       |   |   |   |  |  |  |  |  |  |
|-------|---|---|---|--|--|--|--|--|--|
| SUPAX | 4 | 3 | 2 |  |  |  |  |  |  |
|-------|---|---|---|--|--|--|--|--|--|

| Descriptor       | Meaning  |
|------------------|--|
| RID <sub>i</sub> | Ring identification number. (Integer > 0)  |
| HID <sub>i</sub> | Harmonic identification number. (Integer $\geq 0$ )  |
| C <sub>i</sub>   | Conical shell degree-of-freedom numbers. (Any unique combination of the Integers 1 through 6.) |

**Remarks:**

1. SUPAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may appear on a single entry.
3. Degrees-of-freedom appearing on SUPAX entries may not appear on MPCAX, SPCAX, or OMITAX entries.
4. For a discussion of conical shell analysis, see [Conical Shell Element \(RINGAX\)](#) (p. 145) in the *MSC Nastran Reference Guide*.

**SUPPORT**

## Fictitious Support

Defines determinate reaction degrees-of-freedom in a free body.

**Format:**

| 1       | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|---------|-----|----|-----|----|-----|----|-----|----|----|
| SUPPORT | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|         |    |     |  |  |  |  |  |  |  |
|---------|----|-----|--|--|--|--|--|--|--|
| SUPPORT | 16 | 215 |  |  |  |  |  |  |  |
|---------|----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

**Remarks:**

1. The SUPPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint (i.e., SPCi entry or PS on the GRID entry).
2. SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.
3. Be careful not to spell SUPPORT with two Ps.
4. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
5. From 1 to 24 support degrees-of-freedom may be defined on a single entry.
6. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
7. An alternative to SUPPORT is the SUPPORT1 entry, which is requested by the SUPPORT1 Case Control command.
8. The SUPPORT entry is not allowed in SOLs 106, 129, and 400.
9. The SUPPORT entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD. Therefore, SOL101 runs with contact should be converted over to SOL400.

**SUPPORT1**

## Fictitious Support, Alternate Form

Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPPORT1 must be requested by the SUPPORT1 Case Control command.

**Format:**

| 1        | 2   | 3   | 4      | 5   | 6  | 7   | 8  | 9 | 10 |
|----------|-----|-----|--------|-----|----|-----|----|---|----|
| SUPPORT1 | SID | ID1 | C1     | ID2 | C2 | ID3 | C3 |   |    |
|          | ID4 | C4  | -etc.- |     |    |     |    |   |    |

**Example:**

|          |   |    |     |  |  |  |  |  |  |
|----------|---|----|-----|--|--|--|--|--|--|
| SUPPORT1 | 5 | 16 | 215 |  |  |  |  |  |  |
|----------|---|----|-----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Identification number of the support set. See Remark 1. (Integer > 0)  |
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

**Remarks:**

1. The SUPPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint; (i.e., SPCi entry or PS on the GRID entry).
2. SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.
- In SOL 101, PARAM,INREL,-1 must also be specified or the SUPPORTi entries will be treated as constraints.
3. SUPPORT1 must be requested by the SUPPORT1 Case Control command. The degrees-of-freedom specified on SUPPORT1 will be combined with those on the SUPPORT entry.
4. Be careful not to spell SUPPORT with two Ps.
5. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
6. From 1 to 18 support degrees-of-freedom may be defined on a single entry.
7. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
8. In superelement analysis, SUPPORT1 may be specified for points belonging to the residual structure only.

9. The SUPPORT1 entry is not allowed in SOLs 106, 129, and 400.
10. The SUPPORT1 entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. *Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD.* Therefore, SOL101 runs with contact should be converted over to SOL400.

**SUPPORT6**

## Inertia Relief for SOL 600

Inertia relief used in SOL 600 only.

**Format:**

| 1        | 2   | 3    | 4      | 5 | 6 | 7 | 8    | 9 | 10 |
|----------|-----|------|--------|---|---|---|------|---|----|
| SUPPORT6 | SID | METH | IREMOV |   |   |   | IDS1 |   |    |

**Example:**

|          |   |   |    |  |  |  |     |  |  |
|----------|---|---|----|--|--|--|-----|--|--|
| SUPPORT6 | 0 | 3 | 1  |  |  |  | 101 |  |  |
| SUPPORT6 | 4 | 3 | -2 |  |  |  |     |  |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Set ID corresponding to a Case Control SUPPORT1 entry or zero. (Integer; Default = 0)   |
| 0          | If this is the only SUPPORT6 entry, use this SUPPORT6 entry for all subcases.<br>If there are multiple SUPPORT6 entries, use the one with SID=0 for Marc increment zero.                        |
| N          | Use this SUPPORT6 entry for the subcase specified by Case Control SUPPORT1=N.   |
|            | Different SUPPORT6 entries can be used for each subcase if desired and different subcases can use different methods.  |
|            | If there is only one SUPPORT6 entry (with SID=0), no Case Control SUPPORT1 entries are necessary.   |
| METH       | Method to use (Integer; Default = 0)  |
| 0          | Inertia relief is not active for this subcase.  |
| 3          | Use the “Support Method”, usually specified using param,inrel,-1 for other solution sequences. (See Remark 1.) Input will come from all SUPPORT entries and those SUPPORT1 entries with ID=SID. |
| IREMOV     | Method to retain or remove inertia relief from a previous subcase (Integer; Default = 1).   |
| 1          | Retain inertia relief conditions from previous subcase.   |
| -1         | Remove inertia relief loads immediately.  |
| -2         | Remove inertia relief loads gradually   |

| Descriptor | Meaning  |
|------------|--|
|            | IREMOV should be blank or 1 unless METH is 0.  |
| IDS1       | ID of SUPORT1 entries to be used if METH=3 and SID=0 (Integer; no Default).<br><br>For METH=3, only SUPPORT1 entries with ID=IDS1 will be used in Marc increment zero. All SUPPORT entries will be used.<br><br>(Used for METH=3 when SID=0 ONLY.) |

**Remark:**

1. The parameter INREL is ignored by SOL 600.

**SURFINI****Eulerian Initialization Surface**

Defines a surface that is used for initialization of regions of an Eulerian mesh. Used in SOL 700 only.

**Format:**

| 1       | 2   | 3    | 4     | 5       | 6     | 7 | 8 | 9 | 10 |
|---------|-----|------|-------|---------|-------|---|---|---|----|
| SURFINI | VID | BSID | COVER | REVERSE | CHECK |   |   |   |    |

**Example:**

|         |     |    |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|
| SURFINI | 100 | 37 |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning   |  |
|------------|---|--|
| VID        | Unique number of an SURFINI region entry. Referenced from TICEUL1. (Integer > 0; Required)                                    |  |
| BSID       | ID of a BSURF entry defining the initialization surface. (Integer > 0; Required)  |  |
| COVER      | The processing strategy for Eulerian elements inside and outside of the initialization surface. (Character; Default = INSIDE) |  |
|            | INSIDE  | The part of the Eulerian elements that lie inside the closed volume of the initialization surface will obtain the initial conditions belonging to that surface.              |
|            | OUTSIDE   | The part of the Eulerian elements that lie outside the closed volume of the initialization surface will obtain the initial conditions that belong to that surface            |
| REVERSE    | Auto reverse switch for SURFINI surface segments. (Character; Default = ON)   |  |
|            | ON  | If necessary, the normals of the SURFINI surface segments are automatically reversed so that they all point in the same general direction and give a positive closed volume. |
|            | OFF   | The segments normals are not automatically reversed.   |
| CHECK      | Checking switch for SURFINI surface segments. (Character; Default = ON)   |  |
|            | ON  | The normals of the segments are checked to see whether they all point in the same general direction and give a positive closed volume.                                       |
|            | OFF   | The segment normals are not checked.   |
|            | When "REVERSE" is set to "ON", "CHECK" is automatically set to "ON".  |  |

**Remarks:**

1. All initialization surfaces must form a multifaceted closed volume.

2. An initialization surface can only be used to initialize regions in a Eulerian mesh with appropriate initial conditions. An initialization surface cannot be used as a coupling surface, contact surface or rigid surface.
3. The normal vectors of all segments that form the initialization surface must point in the same general direction, and result in a positive, closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of how segments are initially defined.
4. The “COVER” option determines how Eulerian elements that are (partially) inside or outside of the initialization surface are processed.

**SWLDPRM**

Parameters for CFAST, CWELD, and CSEAM Connector Elements

Overrides default values of parameters for CFAST, CWELD, and CSEAM connectivity search.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| SWLDPRM | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 | PARAM4 | VAL4 |    |
|         | PARAM5 | VAL5 | -etc.- |      |        |      |        |      |    |

**Example:**

|         |        |      |        |   |       |   |  |  |  |
|---------|--------|------|--------|---|-------|---|--|--|--|
| SWLDPRM | GSPROJ | 15.0 | GSMOVE | 2 | PRTSW | 1 |  |  |  |
|---------|--------|------|--------|---|-------|---|--|--|--|

**Alternate Format and Examples:**

| 1       | 2      | 3     | 4      | 5     | 6      | 7     | 8      | 9    | 10 |
|---------|--------|-------|--------|-------|--------|-------|--------|------|----|
| SWLDPRM | PARAM1 | VAL1  | PARAM2 | VAL2  | etc.   | CFAST | PARAM1 | VAL1 |    |
|         | PARAM2 | VAL2  | etc.   | CWELD | PARAM1 | VAL1  | PARAM2 | VAL2 |    |
|         | etc.   | CSEAM | PARAM1 | VAL1  | PARAM2 | VAL2  | etc.   |      |    |

|         |         |        |       |         |     |       |   |       |  |
|---------|---------|--------|-------|---------|-----|-------|---|-------|--|
| SWLDPRM | CHKRUN  | 2      |       |         |     |       |   |       |  |
|         | CWELD   | GSMOVE | 2     | PROJTOL | .04 | PRTSW | 1 | CSEAM |  |
|         | PROJTOL | 0.06   | GMCHK | 1       |     |       |   |       |  |

|         |        |         |      |         |     |       |   |  |  |
|---------|--------|---------|------|---------|-----|-------|---|--|--|
| SWLDPRM | CHKRUN | 2       |      |         |     |       |   |  |  |
|         | CWELD  | GSMOVE  | 2    | PROJTOL | .04 | PRTSW | 1 |  |  |
|         | CSEAM  | PROJTOL | 0.06 | GMCHK   | 1   |       |   |  |  |

| Descriptor                | Meaning  |
|---------------------------|--|
| PARAMi                    | Name of the connector parameter. Allowable names are listed in <a href="#">Table 9-36</a> . (Real or Integer)  |
| VALi                      | Value of the parameter. See <a href="#">Table 9-36</a> (Real or Integer)   |
| CFAST,<br>CWELD,<br>CSEAM | Keywords to control element type specific parameters. Any parameter following a keyword is applied only to that element type. See Remarks 2 and 3. (Character) |

Table 9-36 PARAMi Names and Descriptions

| Name                 | Type                                    | Default | Description  |
|----------------------|---|---------|--|
| ACTVTOL<br>(SOL 600) | Integer $\geq 0$<br>Integer $\leq 2211$ | 1111    | <p>Parameter controlling the behavior of PROJTOL for the different CWELD connection methods. This parameter is entered as an integer and is converted to a four-character string. If its value is less than 1000, the string will be prepended with zeros. The first character (from the left) controls the behavior when method PARTPAT is used. The second controls the behavior when method ELPAT is used. The third controls the behavior when method ELEMID is used and the fourth controls the behavior when method GRIDID is used.</p> <p>For ALIGN the PROJTOL tolerance has no significance. Each digit (<math>d_i</math>) in the string can have the value 0 or 1 or 2, where the value 2 only has significance for the methods ELPAT or PARTPAT. The values have the following meaning:</p> <ul style="list-style-type: none"> <li>0 = PROJTOL is completely deactivated</li> <li>1 = PROJTOL is activated for ELEMID and GRIDID, PROJTOL is activated in initial projections for ELPAT, PROJTOL is only activated over free edges of the patch in auxiliary projections for ELPAT and in initial and auxiliary projections for PARTPAT. Free edges have no neighbors within the set that defines the complete surface.</li> <li>2 = PROJTOL is always activated</li> </ul> |
| CHKRUN               | Integer 0, 1, 2                         | 0       | <p>Stop run or allow run to continue after the connectivity elements are generated.</p> <p>0=abort on first error;<br/>1=stop after connectivity has been checked; 2=continue run if no errors are found.</p>  |
| CNRAGLI              | 90.0 $\leq$ Real $\leq$ 180.0 or -1.0   | 160.0   | CSEAM only. Minimum angle allowed between the free edges of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is less than the value of CNRAGLI. If set to -1.0, the check will be skipped.   |
| CNRAGLO              | 0.0 $\leq$ Real $\leq$ 90.0 or -1.0     | 20.0    | CSEAM only. Maximum angle allowed between the normal vectors of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is greater than the value of CNRAGLO. If set to -1.0, the check will be skipped.  |

Table 9-36 PARAMi Names and Descriptions

| Name                | Type                     | Default | Description  |
|---------------------|--------------------------|---------|--|
| CSVOUT              | Integer $\geq 0$         | 0       | <p>Print diagnostic output in a comma separated .csv file useful for reports. The users may view or manipulate data using Microsoft Excel, or other spread sheet programs that can process CSV format.</p> <p>0      no .csv output.</p> <p>unitnum &gt; 0 unit number of the .csv file assigned via the ASSIGN statement in the File Management Section, for example,</p> <p>ASSIGN USERFILE=myfile.csv UNIT=<i>unitnum</i><br/>FORM=FORMATTED DELETE STATUS=NEW</p>  |
| CWSETS<br>(SOL 600) | Integer $> 0$            | 0       | <p>Parameter to control the automatic creation of four element sets with the elements involved in the CWELD connections.</p> <p>0 = the sets will not be created</p> <p>1 = four sets will be created automatically<br/>“fastener_all_beams_inc0000”, the set containing all connector beam elements.<br/>“fastener_all_faces_sidea_inc0000”, the set containing all elements with patches on side B of the connection.<br/>“all_fastener_warnings_inc0000”, the set containing all elements involved in CWELD warning messages.</p> <p>Defining sets with any of these names must be avoided and will be considered an error.</p> |
| CWSPOT<br>(SOL 600) | $0 < \text{Integer} < 3$ | 1       | <p>Parameter to choose the method for modifying the beam length.</p> <p>1 = scale the stiffness of the beam<br/>2 = reposition the end nodes of the beam<br/>3 = reposition the auxiliary patch nodes and dthe end nodes of the beam.</p>  |
| DELMAX<br>(SOL 600) | Real                     | 0.1     | Maximum allowable parametric coordinate change during the iteration process for finding the projection on a patch. At first DELMAX is not activated, i.e., the parametric coordinate change is not limited during the iteration process. The parameter is only activated when the full Newton Raphson iteration process for a projection did not converge. In that case the iteration process is restarted with DELMAX activated.  |
| DLDMAX<br>(SOL 600) | Real $> 0.0$             | 5.0     | Default value for LDMAX, the largest ratio of length to characteristic diameter.   |

Table 9-36 PARAMi Names and Descriptions

| Name                | Type               | Default | Description  |
|---------------------|--------------------|---------|--|
| DLLMIN<br>(SOL 600) | Real > 0.0         | 0.2     | Default value for LDMIN, the smallest ratio of length to characteristic diameter.  |
| DRATIO              | 1.0 ≤ Real ≤ 10.0  | 1.0     | CFAST or CWELD (PARTPAT and ELPAT) only. Increases the Connector patch diameter and is used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. See Remark 10.  |
| EPSITR<br>(SOL 600) | Real > 0.0         | 1.0E-5  | Tolerance to terminate the iteration process for finding the projection on a patch. If the parametric coordinate change in an iteration is less than EPSITR the projection is accepted as converged.   |
| GMCHK               | Integer 0, 1, 2, 3 | 0       | <p>For CSEAM, CWELD with ELPAT or PARTPAT format, and CFAST only.</p> <p>0=no geometry error checks;</p> <p>1=check errors of the CSEAM across a cutout or over a corner with patch elements in plane or out of plane;</p> <p>2=check errors of CSEAM and output all candidate shell elements if an error is encountered.</p> <p>If GMCHK=1 or 2 and an error is detected, the program will loop back to search for next candidate element until a good pair of connection is found or all adjacent elements have been checked. In the latter case, a user fatal message 7595, 7638, or 7667 will be issued. A UFM 7595 is issued if the normal angles between the patches at end GS or the patches at end GE exceed the value of GSPROJ; a UFM 7638 is issued if either the length of the seam spans more than three elements or the seam spans a cutout; a UFM 7667 is issued if the normal angles between the top patches at GS and GE or the normal angles between the bottom patches at GS and GE exceeds CNRAGLO or if the angle between the free edges of the shell elements onto which GS and GE are projected is less than CNRAGLI.</p> <p>3=check backward projections for CWELD with ELPAT or PARTPAT format and CFAST. See Remark 8.</p> |

Table 9-36 PARAMi Names and Descriptions

| Name                | Type                                     | Default                     | Description  |
|---------------------|--|-----------------------------|--|
| GSMOVE              | Integer                                  | 0                           | Maximum number of times GS for the CFAST or CWELD (PARTPAT or ELPAT options only) or GS/GE for the CSEAM is moved in case a complete projection of all auxiliary points has not been found. See Remark 9.  |
| MOVGAB              | Integer 0, 1                             | 0                           | <p>Option to correct the locations of user defined GA/GB for CFAST and CWELD elements.</p> <p>0 = keep the locations of the user specified GA/GB and connect them to shell elements directly.</p> <p>1 = generate new grids with corrected locations to connect shell elements.</p>  |
| GSPROJ              | $0.0 \leq \text{Real} \leq 90.0$ or -1.0 | 20.0<br>(-89.0 for SOL 600) | Maximum angle allowed between the normal vectors of shell A and shell B. The connector element will not be generated if the angle between these two normal vectors is greater than the value of GSPROJ. For CSEAM, see also GMCHK for additional error checks using GSPROJ. If GSPROJ is set to -1.0 (or -89.0 for SOL 600), the program will skip the checking of GSPROJ.   |
| GSTOL               | $\text{Real} \geq 0.0$                   | 0.0                         | For CFAST or CWELD (PARTPAT and ELPAT only), if GSTOL > 0.0 and the distance between GS and the projected point GA or GB is greater than the GSTOL, a UFM 7549 is issued and the CFAST or CWELD is rejected. For CSEAM, if GSTOL > 0.0 and the distance between GS and the projected point GSA or GSB or the distance between GE and the projected point GEA or GEB is greater than the GSTOL, a UFM 7549 is issued and the CSEAM is rejected.   |
| MAXEXP<br>(SOL 600) | Integer > 0                              | 2                           | Parameter to control the maximum number of expansions in the search for projections of the auxiliary nodes. First the master patch will be searched. If no projection is found on the master patch a first expansion will be made including all neighboring patches of the master patch. If no projection is found on any of the new patches a second expansion will be made including all neighbors of the patches tried so far. This process continues until the number of expansions exceeds MAXEXP. Two patches are neighbors if they share at least one node in their connectivities. |

Table 9-36 PARAMi Names and Descriptions

| Name                | Type                                 | Default                     | Description  |
|---------------------|--------------------------------------|-----------------------------|--|
| MAXITR<br>(SOL 600) | Integer > 0                          | 20                          | The maximum number of iterations allowed in the iteration process for finding the projection on a patch.   |
| NREDIA              | Integer 0, 1, 2, 3,<br>4, 5, 6, 7, 8 | 0                           | CFAST or CWELD (PARTPAT and ELPAT) only. Maximum number of times the diameter D is reduced in half in case a complete projection of all points has not been found.   |
| PROJTOL             | $0.0 \leq  \text{Real}  \leq 1.0$    | 0.02<br>(0.199 for SOL 600) | <p>For CFAST or CWELD, tolerance to accept the projected point GA or GB if the computed coordinates of the projection point lie outside the shell element but is located within <math> \text{PROJTOL}  \times (\text{dimension of the shell element forming the patch})</math>. For the CSEAM, a projection from GS/GE will always be attempted as if PROJTOL=0.0 and if one cannot be found then the non-zero value of <math> \text{PROJTOL} </math> will be used.</p> <p>Regardless of the value of SWLDPRM PROJTOL, the algorithm starts by assuming a zero projection tolerance for the projections of GA/GB for the CWELD option "PARTPAT" or the CFAST option "PROP" and for GAHi/GBHi for the CWELD options "PARTPAT" and "ELPAT" and any CFAST option. The tolerance is increased by 0.02 until a projection is found or the PROJTOL value is reached.</p> <p>This can be turned off while computing the auxiliary grid projection onto EIDA/EIDB or onto the adjacent elements of EIDA/EIDB by setting PROJTOL= - <i>value</i> where <math>0.0 \leq \text{value} \leq 1.0</math>. In this case, the projection calculation starts at tolerance = <math> \text{PROJTOL} </math>. For the rest of the projection search, the algorithm reverts back to the iteration starting at tolerance = 0.0.</p> |
| WMASS               | Integer 0, 1                         | 0                           | For CWELD to react to a non-zero density 1 value is required.  |

Table 9-36 PARAMi Names and Descriptions

| Name                | Type                                    | Default           | Description  |
|---------------------|---|-------------------|--|
| PRTSW               | Integer 0, 1, 2 , 11, 12, 100, 101, 111 | 0 (2 for SOL 600) | <p>Print diagnostic in output or punch out internally generated RBE3 elements and auxiliary grids in Bulk Data format for the connector elements.</p> <p>0=no diagnostic output (appropriate USER MESSAGES such as 'USER WARNING MESSAGE 7636 will continue to be issued);</p> <p>1=print diagnostic output in exponential format to f06 file;</p> <p>2=punch diagnostic output in exponential format to .pch file;</p> <p>11=print diagnostic output in real format to .f06 file;</p> <p>12=punch diagnostic output in real format to .pch file.</p> <p>100=punch out Bulk Data without diagnostic output.</p> <p>101= punch out Bulk Data and print diagnostic output in exponential format to f06 file.</p> <p>111= punch out Bulk Data and print diagnostic output in real format to f06 file.</p> |
| RBE3WT<br>(SOL 600) | Real                                    | 0.0               | <p>Default RBE3 distance weighting exponent.</p> <p>The weight factor for each retained node in a RBE3 involved in a CWELD connection is: <math>f_i = 1/d_i^n</math>, where:</p> <p><math>f_i</math> is the weighting factor for retained node 1.</p> <p><math>d_i</math> is the distance from the tied node to retained node i</p> <p><math>n</math> is the weighting exponent RBE2WT</p> <p>Negative values for RBE3WT are not recommended, since they will result in heavier weighting for nodes further away. The default results in uniform weighting (<math>f_1 = 1</math>).</p>   |

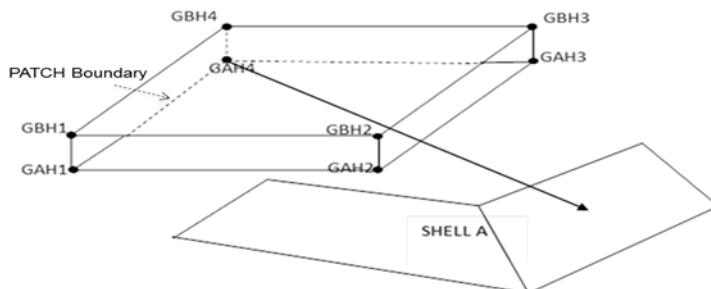
Table 9-36 PARAMi Names and Descriptions

| Name                | Type            | Default | Description   |
|---------------------|-----------------|---------|---|
| GSCURV<br>(SOL 600) | -90 < Real < 90 | 20.0    | <p>Maximum angle allowed between the normal vectors of a patch to which an auxiliary node projects and its corresponding auxiliary and master patches. It provides a measure to monitor the curvature of a surface and to recognize patches that belong to, for example, stiffeners. A connection is not generated if the angle between the normal vectors is greater than 90-GSCURVE meaning that the patches are almost normal to each other. In that case, the patch is reflected and the search proceeds to the next patch in the list. If the angle is between zero and GSCURV, no message is displayed. If the angle is between GSCURV and 90-GSCURV, a large angle warning is displayed. The following three tests are performed in the order given below when GSCURV is positive:</p> <p>If <math>0 &lt; \text{angle} &lt; \text{GSCURVE} \geq \text{OK}</math><br/> If <math>\text{GSCURVE} &lt; \text{angle} &lt; 90-\text{GSCURV} &gt;</math> trigger a warning.<br/> If <math>\text{angle} &gt; 90-\text{GSCURV} \geq \text{reject}</math>.</p> <p>Note that the warning condition is never triggered when <math>\text{GSCURV} &gt; 45</math> as it is overruled by the reject condition.</p> <p>If GSCURV is negative, the projection is always accepted and a warning is issued when the angle is larger than <math> \text{GSCURV} </math>.</p> |
| SCLSKIN             | Real $\geq 0.0$ | 0.0     | CFAST or CWELD (PARTPAT and ELPAT) only. If set to a value $> 0.0$ the CFAST or CWELD connector will be stiffened by increasing the bending moment of inertia ratio by SCLSKIN of the shell elements involved in the connector patch. SCLSKIN is Factor used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. An initial recommended value is 0.1; See Remark 10..  |

**Remarks:**

1. This entry changes the default settings of control variables for the CFAST, CWELD, and CSEAM connector elements. None of the parameters of this entry are required. Only one SWLDPRM entry is allowed in the Bulk Data Section.

2. If any of the key words CFAST, CWELD, and CSEAM does not appear on this entry, then a parameter set on this entry is considered "global" and applies to all the connector elements in the model. Any parameter set on this entry that comes before a key word CFAST, CWELD, or CSEAM is considered global.
3. Any parameter set on the entry that comes after a key word such as CFAST will only apply to that connector element type until another key word such as CSEAM is encountered. If a parameter is defined for a specific connector type that does not apply to that connector type then it will be ignored.
4. Blank fields are allowed for readability. However, a parameter name must be followed in the immediately following field by the corresponding parameter value. If the parameter name falls in the field just before a continuation field, then its parameter value must be placed in the first field after the continuation marker of the continuation entry.
5. Connectivity information is generated for the CFAST and CSEAM elements. For the CWELD elements, connectivity information is only generated for the PARTPAT, ELPAT, ELEMID, and GRIDID options.
6. The details of individual connector connectivity can be found on the appropriate CFAST, CWELD, and CSEAM Bulk Data entries.
7. The CHKRUN parameter must be global.
8. Backward connections sometimes occur if the patch is near the boundary of a structure and there is a "vertical" flange associated with the patch elements. In this case GMCHK=3 may be used to prevent backward projection. See the figure below. If GMCHK=3 and a backward projection is detected, the program will not connect this projected shell element. Instead, it will continue searching iterations until a satisfying connection is reached.

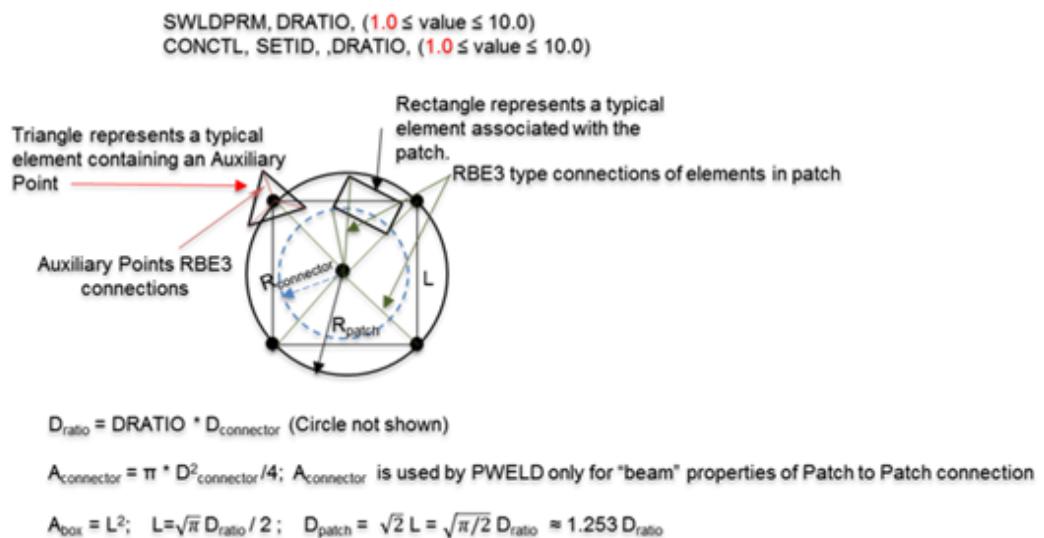


9. If the GMOVE specification limit is reached for the CFAST or the CWELD with options "PARTPAT" and "ELPAT" and SWLDPRM NREDIA  $\geq 0$ ; then the diameter of the connector will be reduced by half to compute new locations of auxiliary grids. If necessary this is repeated until the NREDIA specified value is reached.
  - a. When the NREDIA  $\neq 0$  is initiated, the GS at its current location is used for GMOVE  $\geq 0$ .
  - b. When the NREDIA  $\neq 0$  is initiated, the GS at its original location is used for the new option GMOVE  $< 0$ .

10. Connector contribution to a structural model's overall stiffness is sensitive to the model's mesh size and the orientation of the connector relative to the mesh. Thus, the discretization process itself may cause, for example, a model using a fine mesh to be stiffer in torsion than a corresponding model using a coarse mesh. Also for production models that correlate well with test, refining the mesh may cause an inherent overall loss of stiffness due to mesh refinement and hence loss of correlation.

To allow the user some control over model stiffness, the new connectors (CWELD with ELPAT or PARTPAT or CFAST) are provided with two options to provide additional connector stiffness. The two options may be used individually or in combination.

The first stiffening technique is activated by "SWLDPRM, DRATIO, (1.0 ≤ value ≤ 10.0)" or "CONCTL, SETID, ,DRATIO, (1.0 ≤ value ≤ 10.0)". For this option the diameter,  $D_{ratio}$ , is defined as  $D_{ratio} = DRATIO * D_{connector}$ . This results in the diameter of the patch taking a value of  $D_{patch} = \sqrt{\pi/2} D_{ratio}$ . The default of DRATIO is a value=1.0 which implies the diameter of the patch is computed in the standard fashion. For the patch to patch connection for the "beam" properties of the CWELD, the area is still computed as  $A_{connector} = \pi D_{connector}^2 / 4$  as defined in the PWELD entry.



A *disadvantage* of this method is that as DRATIO is increased using the global command SWLDPRM, DRATIO, *value*; some connector elements may begin to fail because they may no longer be able to find a patch projection.

To overcome this, the "SWLDPRM, NREDIA, Integer\_value" can be increased to a value as high as *Integer\_value* = 8 to allow failing welds to halve their patch diameters up to eight times. Whenever a connector has its diameter reduced a message such as: "DA IS REDUCED BY HALF TO 8.8623E+00" when PRTSW diagnostic output is requested. In this message the DA is computed as : DA = L/2.

If the "SWLDPRM, NREDIA, *Integer\_value*" is not an approach the user wishes to pursue, then for these failing elements, the bulk data entry CONCTL, SETID, ,DRATIO, value can be used to define a set for failing connectors and set a value of DRATIO for these connectors that allows them to find a projection.

The second stiffening algorithm attempts, based on the diameter of the connector, to determine a measure of the mesh discretization.

This feature is activated by setting "SWLDPRM, SCLSKIN > 0.0" or "CONCTL, SETID, SCLSKIN > 0.0". The default is a value = 0.0 which implies no stiffening. When activating the "SWLDPRM, SCLSKIN, real value" option for the first time with a new FEM, it is recommended that an initial value = 0.1 be used.

Depending on the complexity of the model and the overall mesh size and the number of connectors within the model and the diameter of the connectors relative to the mesh, the *default value* tends to stiffen a structural model from about 0.4% to about 4%. A value of SCLSKIN=10.0 stiffens coarser mesh models by about 10% to 11% and finer mesh models by about 2% to 6%.

The contribution of the stiffening algorithm to the overall stiffness of the FEM model eventually reaches a limit. For example, a very large value SCLSKIN=100 increases the stiffness of the models overall by only about 0.1% to 2% over the stiffness obtained for SCLSKIN=10.

For a correlated structural model evaluated at a specific mesh size, *with an aim to refine the mesh for some portion of this model containing connectors*, while leaving other portions containing connectors with an unmodified mesh, it is recommended that the "SCLSKIN, *real value*" be entered on the CONCTL bulk data entry referring to the connectors within the area of the refined mesh. Different refined mesh areas within the structural model can have different values of SCLSKIN associated to the specific connectors in each refined region.

For post processing affected shell elements, an updated EPT table is available after module MODGM2. It contains the PSKNSHL record that correlates the property data of the shells involved and a list of shell elements for each patch modified.

Neither of these stiffening methods have any effect on the mass computations of the model.

11. This entry is ignored in part super-elements. Inside the part super-elements, the default settings will be applied.
12. If Modules are present then this entry may only be specified in the main Bulk Data section.

## Entries T-Y

### TABD1MD

TABLED1 entries internal modification in SOL 600

Defines how TABLED1 entries are internally modified in SOL 600.

SOL 600 usually requires that the first point of all TABLED1 entries used to describe time histories start with time/amplitude of (0.0, 0.0). If the user omits this entry, all TABLED1 entries that do not start with (0.0, 0.0) will be modified internally to add two points at the start, the first at (0.0, 0.0) and the second at (0.001, V1) where V1 is the first amplitude of the original TABLED1 entry for that curve.

#### Format

| 1       | 2    | 3    | 4   | 5   | 6  | 7  | 8  | 9 | 10 |
|---------|------|------|-----|-----|----|----|----|---|----|
| TABD1MD | TID1 | THRU | ID2 | MOD | T2 | A2 | T3 |   |    |

#### Example:

|         |   |      |   |   |      |     |      |  |  |
|---------|---|------|---|---|------|-----|------|--|--|
| TABD1MD | 1 | THRU | 5 | 1 | .002 | 1.0 | .003 |  |  |
|---------|---|------|---|---|------|-----|------|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID1        | First TABLED1 ID. (Integer > 0; no Default required)   |
| ID2        | Last TABLED1 ID to which MOD, T1 and A1 apply. See Remark 2. (Integer; Default = ID1)  |
| MOD        | Flag indicating whether or not to modify TABLED1 entries with ID in the range ID1 to ID2. (Integer; Default = 1) <ul style="list-style-type: none"> <li>0      Do not modify the table</li> <li>1      Modify the table</li> </ul> |
| T1         | New time for the second point. See Remark 1. (Real; Default = 0.001)   |
| A1         | New amplitude of the second point. See Remark 2. (Real; Default = 1.0)   |
| T2         | New time for the third point. See Remark 1. (Real; Default = 0.002)  |

#### Remarks:

- If MOD=1, all TABLED1 entries with ID's in described by ID1 to ID2 will be modified to add two points to the beginning of the each table. The first point will be at (0.0, 0.0) the second point will be at (T1,A1). The original first point will be modified to be at time T3.
- All TABLED1 entries in the range ID1 to ID2 will be modified as indicated. Repeat this entry as many times as necessary to specify all TABLED1 ID's that should either be modified or not be modified.
- Make sure to set MOD=0 to all TABLED1 entries that are not used to describe time history loading.
- The ID1 to ID2 range may include values that do not have any TABLED1 ID's in the model.

5. If all TABLED1 entries are to be modified or if they do not all start with the same T2, A2, T3 values, PARAM,MTALBD1M PARAM,MTALBD1T may be entered.

**TABDMP1**

## Modal Damping Table

Defines modal damping as a tabular function of natural frequency.

**Format:**

| 1       | 2   | 3    | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|------|----|----|----|----|--------|---|----|
| TABDMP1 | TID | TYPE |    |    |    |    |        |   |    |
|         | f1  | g1   | f2 | g2 | f3 | g3 | -etc.- |   |    |

**Example:**

|         |     |        |     |        |      |  |  |  |  |
|---------|-----|--------|-----|--------|------|--|--|--|--|
| TABDMP1 | 2   |        |     |        |      |  |  |  |  |
|         | 2.5 | .01057 | 2.6 | .01362 | ENDT |  |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. (Integer > 0)                              |
| TYPE       | Type of damping units. (Character: "G", "CRIT", or "Q"; Default is "G") |
| fi         | Natural frequency value in cycles per unit time. (Real $\geq 0.0$ )     |
| gi         | Damping value. (Real)   |

**Remarks:**

1. Modal damping tables must be selected with the Case Control command SDAMPING = TID.
2. The frequency values, fi, must be specified in either ascending or descending order, but not both.
3. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-159](#) discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-159](#), the value of g at f = f3 is  $g = (g_3 + g_4)/2$ .
4. At least one continuation entry must be specified.
5. Any fi or gi entry may be ignored by placing "SKIP" in either of the two fields used for that entry.
6. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
7. The TABDMP1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. The table look-up  $g_T(f)$  is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points. See [Figure 9-159](#). No warning messages are issued if table data is input incorrectly. See Remark 11.

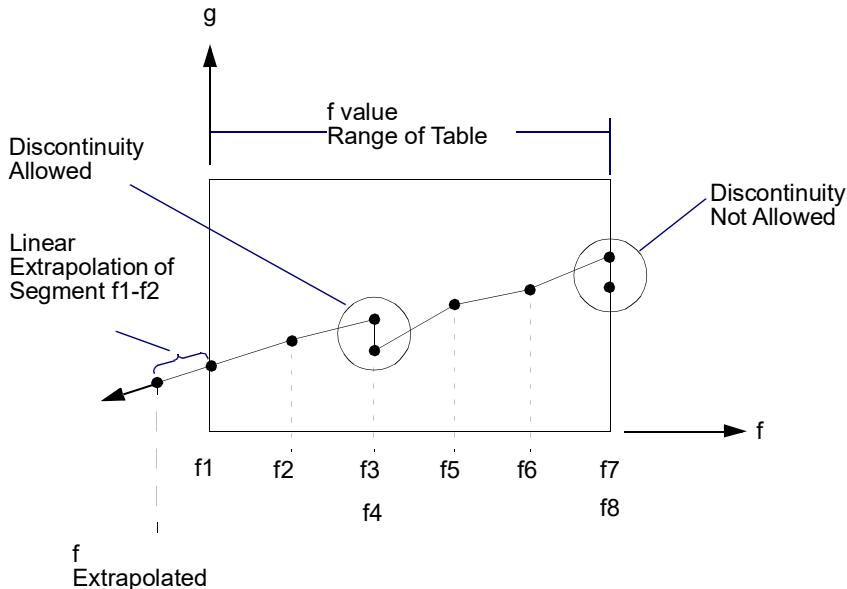


Figure 9-159 Example of Table Extrapolation and Discontinuity

8. This form of damping is used only in modal formulations of complex eigenvalue analysis, frequency response analysis, or transient response analysis. The type of damping used depends on the solution sequence (structural damping is displacement-dependent, and viscous damping is velocity-dependent). See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for the equations used.
9. PARAM,KDAMP may be used in solution sequences that perform modal frequency and modal complex analysis, to select the type of damping.

| KDAMP       | Result      |
|-------------|-------------|
| 1 (Default) | $B$ Matrix  |
| -1          | $(1 + ig)K$ |

See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for a full explanation.

10. If TYPE is "G" or blank, the damping values  $g_i$ , etc., are in units of equivalent viscous dampers, as follows:

$$b_i = \frac{g_i}{\omega_i} K_i$$

(See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*) If TYPE is “CRIT”, the damping values  $g_i$ , etc., are in the units of fraction of critical damping  $C/C_0$ . If TYPE is “Q”, the damping values  $g_i$  are in the units of the amplification or quality factor,  $Q$ . These constants are related by the following equations:

$$C/C_0 = g/2$$

$$Q = \begin{cases} 1/(2C/C_0) \\ 1/g \end{cases}$$

11. A user warning message is used if either of the following conditions is satisfied:
  - a. The modal damping value is computed as a result of extrapolation.
  - b. The computed modal damping value is negative.

For any modal damping value that satisfies condition a or b, the program lists the cyclic frequency and the corresponding modal damping value and indicates whether this value was computed as a result of interpolation or extrapolation. For the latter case, it also indicates whether the extrapolation was beyond the left end of the table or beyond the right end of the table.

If a modal damping value satisfies *both* of the conditions, a and b above (that is, the modal damping value is computed as a result of extrapolation *and* it is negative), the program terminates the job with a user fatal message.

The user can prevent the program from terminating the job as above by specifying MDAMPEXT=1 [or SYSTEM(426)=1] on the NASTRAN statement. The user fatal message mentioned above does inform the user of this avoidance scheme.

12. The modal damping matrix generated by this approach is not affected by the presence of rotors in the model. Also, the circulation terms are not generated for damping corresponding to rotor degrees of freedom.
13. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TABL3D****Multi-Dimensional Table**

Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc for SOL 600 and SOL 400.

**Format 0: Simple Table Type 0 - entry is a function of only one variable:**

| 1       | 2    | 3  | 4      | 5     | 6     | 7     | 8  | 9  | 10 |
|---------|------|----|--------|-------|-------|-------|----|----|----|
| TABL3D0 | ITID |    | KIND   | EXTRP | ITIDS | ITIDB | SM |    |    |
|         | X1   | Y1 | X2     | Y2    | X3    | Y3    | X4 | Y4 |    |
|         | X5   | Y5 | -etc.- |       |       |       |    |    |    |

**Format 1: Multi-Dimensional Table Type 1 - entry is a function of 2, 3, or 4 variables, data entered one row at a time**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10             |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----------------|
| TABL3D1 | ITID   | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |                |
|         | NW3    | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |                |
|         | ITIDS1 | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |                |
|         | SM1    | SM2    | SM3    | SM4    |        |        |        |        |                |
|         | X11    | X12    | X13    | X14    | X15    | X16    | -etc.- |        |                |
|         | X21    | X22    | X23    | X24    | X25    | X26    | -etc.- |        | Enter if NW2>0 |
|         | X31    | X32    | X33    | X34    | X35    | X36    | -etc.- |        | Enter if NW3>0 |
|         | X41    | X42    | X43    | X44    | X45    | X46    | -etc.- |        | Enter if NW4>0 |
|         | Y1     | Y2     | Y3     | Y4     | Y5     | Y6     | -etc.- |        | See Remark 1   |

**Format 2: Multi-Dimensional Table Type 2- entry is a function of 2, 3, or 4 variables, data entered one point at a time**

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10             |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|----------------|
| TABL3D2 | ITID   | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |                |
|         | NW3    | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |                |
|         | ITIDS1 | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |                |
|         | SM1    | SM2    | SM3    | SM4    |        |        |        |        |                |
|         | X11    | X12    | X13    | X14    | X15    | X16    | -etc.- |        |                |
|         | X21    | X22    | X23    | X24    | X25    | X26    | -etc.- |        | Enter if NW2>0 |
|         | X31    | X32    | X33    | X34    | X35    | X36    | -etc.- |        | Enter if NW3>0 |

|  |     |     |     |     |     |     |        |  |                |
|--|-----|-----|-----|-----|-----|-----|--------|--|----------------|
|  | X41 | X42 | X43 | X44 | X45 | X46 | -etc.- |  | Enter if NW4>0 |
|  | Y1  | Y2  | Y3  | Y4  | Y5  | Y6  | -etc.- |  | See Remark 2   |

Format 3: Multi-Dimensional Table Type 3- entry is specified by a formula  
(SOL 600 only)

| 1       | 2       | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10           |
|---------|---------|--------|--------|--------|--------|--------|--------|--------|--------------|
| TABL3D3 | ITID    | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |              |
|         | NW3     | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |              |
|         | ITIDS1  | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |              |
|         | SM1     | SM2    | SM3    | SM4    |        |        |        |        |              |
|         | Formula |        |        |        |        |        |        |        | See Remark 3 |

| Descriptor                       | Meaning   |
|----------------------------------|---|
| ITID                             | Table identification number. (Integer > 0; no Default)  |
| SM or SMi                        | Flag to indicate smoothing of the table data. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0      Do not smooth the data</li> <li>1      Smooth the data</li> </ul>    |
| NV                               | Number of variables the entry is a function of (Integer, 1, 2, 3, or 4; no Default)   |
| KIND or KINDi                    | “Independent” variable type (such as strain, temperature, Integer > 0; no Default; see <a href="#">Table 9-37</a> for application values)   |
| EXTRP or EXTRPi                  | Extrapolation flag. (Integer; Default = 2) <ul style="list-style-type: none"> <li>1      Do not allow extrapolation</li> <li>2      Allow extrapolation (both ends of curve)</li> </ul> |
| ITIDS or ITIDSi, ITIDB or ITIDBi | Table IDs meant for future expansion to reference other tables from this table when trying to evaluate this table outside its defined range. Currently not used, leave blank.           |
| Nwi                              | Number of X values of each variables. (i can range from 1 to 4) (Integer > 0; no Default)   |
| Xi or Xij                        | Value of “independent” variable such as strain, temperature, ... (Real; no Default)   |
| Yi                               | Value of the quantity desired such as stress, Poisson’s ratio, ... (Real; no Default)   |

The “Independent” variable(s) should be selected from [Table 9-37](#):

**Table 9-37**

| Independent Variable Type |  |    |  |    |   |
|---------------------------|--|----|--|----|---|
| 1                         | time   | 26 | $z_0$ coordinate   | 51 | wavelength (used in spectral radiation)               |
| 2                         | normalized time                                    | 27 | $s_0 = \sqrt{x_0^2 + y_0^2 + z_0^2}$   | 52 | creep strain  |
| 3                         | increment number                                   | 28 | contact force $ F $  | 53 | pressure or primary quantity in diffusion             |
| 4                         | normalized increment time                          | 29 | not available  | 54 | equivalent strain rate for non-Newtonian viscosity    |
| 5                         | x coordinate                                       | 30 | $\sigma_n$ (normal stress)   | 55 | normalized arc distance                               |
| 6                         | y coordinate                                       | 31 | voltage**  | 56 | distance to other contact surface (near contact only) |
| 7                         | z coordinate                                       | 32 | current**  | 57 | terms of series                                       |
| 8                         | $s = \sqrt{x^2 + y^2 + z^2}$                       | 33 | $\left(\frac{\text{current radius}}{\text{radius of throat}}\right)^2$<br>(see throat) | 58 | hydrostatic stress                                    |
| 9                         | $\theta$ angle                                     | 34 | Not available  | 59 | hydrostatic strain                                    |
| 10                        | mode number  | 35 | Not available  | 60 | Not available   |
| 11                        | frequency  | 36 | Not available  | 61 | Not available   |
| 12                        | temperature  | 37 | gasket closure distance  | 62 | 2nd state variable                                    |
| 13                        | function   | 38 | displacement magnitude   | 63 | 3rd state variable                                    |
| 14                        | fourier  | 39 | stress rate  | 64 | 4th state variable                                    |
| 15                        | $\bar{\varepsilon}^p$ (equivalent plastic strain)  | 40 | experimental data  | 65 | 5th state variable                                    |
| 16                        | $\dot{\bar{\varepsilon}}$ (equivalent strain rate) | 41 | porosity   | 66 | loadcase number*                                      |
| 17                        | Not available                                      | 42 | void ratio   | 67 | degree of cure*                                       |
| 18                        | arc length   | 43 | $\dot{\bar{\varepsilon}}^c$ (equivalent creep strain rate)                             | 68 | magnetic field intensity**                            |
| 19                        | relative density (not available for shells)        | 44 | minor principal strain   | 69 | equivalent mechanical strain                          |
| 20                        | $\bar{\sigma}$ (equivalent stress)                 | 45 | distance from neutral axis (-1/2, +t/2)  | 70 | 1st strain invariant                                  |
| 21                        | magnetic induction**                               | 46 | normalized distance from neutral axis (-1, +1)   | 71 | 2nd strain invariant                                  |
| 22                        | velocity   | 47 | local x-coordinate of layer point for open or closed section beam                      | 72 | 3rd strain invariant                                  |

Table 9-37

| Independent Variable Type |                      |    |   |    |                           |
|---------------------------|----------------------|----|---|----|---------------------------|
| 23                        | parameter diameter** | 48 | local y-coordinate of layer point for open or closed section beam | 73 | local strain component    |
| 24                        | $x_0$ coordinate     | 49 | not available   | 74 | damage                    |
| 25                        | $y_0$ coordinate     | 50 | not available   | 75 | accumulated crack growth  |
|                           |                      |    |   | 76 | relative sliding velocity |

## Remarks (General):

1. Independent Variable Types marked as 'NOT AVAILABLE' are not available in either SOL 600 or SOL 400. Independent variable types marked with \* are only available in SOL 600. Independent variable types marked with \*\* are only available in SOL 600 through hand-editing of the Marc input file. The rest are available in SOL 600 and SOL 400.
2. Dependent quantities cannot be arbitrary functions of the Independent Variable Types shown in [Table 9-37](#). For e.g., Young's Modulus can be varied with Temperature and Space but cannot be varied with Time. Incorrect tabular settings that are not allowed will be errored out by the program.
3. For time-independent materials, dependent variables (i.e., yield stress) can typically be defined only as a function of temperature, stress/strain data or coordinates. For time-dependent materials, (e.g., creep coefficient), properties can also be defined as a function of time.
4. For contact dependent variables, (i.e., friction coefficient) can typically be defined as a function of temperature, contact body quantities like normal stress, body force, distance from body, relative velocity, etc.
5. If the independent variable is out of range of the table, the user can indicate if the last point in the table should be used or if the table should be extrapolated. Extrapolation means the table is continued with its first or last slope. Care should be taken with extrapolation, particularly for material properties.
6. Independent variable types for coordinates (5, 6, 7) depend on the type of analysis flagged. For a small strain or Total Lagrange or non-mechanical analysis, these coordinates refer to the original coordinates. For Updated Lagrange mechanical analysis, these coordinates refer to the updated
7. At present, options to specify additional tables using ITIDS, ITIDB, ITIDS1, ITIDB1, ... ITIDS4, ITIDB4 are not active and will be ignored if entered.

## Remarks (for SOL 600 only):

1. The function is read by giving NW1 data points ( $NW4^*NW3^*NW2$ ) times. The program reads the data using the following method.

```

do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      read nw1 values f(X1, K2, K3, K4)
    enddo
  enddo
enddo

```

2. The function is read one value at a time. There are NW1\*NW2\*NW3\*NW4 values. The program uses the values as follows:

```

do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      do k1=1, nw1
        read one value f(K1, K2, K3, K4)
      enddo
    enddo
  enddo
enddo

```

3. The formula can extend from field 2 through field 9 and must be comprised of the items listed previously.
4. This parameter may also be used to control the BOLT entry. For BOLT, the default for MMBOLTUS=1 to achieve the same results as SOL 400. If the results appear to be backwards, set MMBOLTUS=-1. BOLT and MBOLTUS may not both be entered in the same model.

#### Remarks for SOL 400 only:

1. Independent variable types are typically specified for materials or contact. When material properties are varied through TABL3DX, the tables are honored only for elements with property extensions. For e.g., if TABL3D0 is used in conjunction with MATEP for a CHEXA mesh with PSOLID properties, then the elements should be given the PSLDN1 property extension for the table to be valid. Note that, with default property mapping (NLMOPTS,SPROPMAP,0), the property extensions will be added automatically based on the rules specified in Remark 9. of the NLMOPTS entry.
2. The strains and stresses identified as independent variables depend on the parameters flagged for large displacement.
  - a. For PARAM,LGDISP,-1, the strains and stresses are engineering quantities.
  - b. For PARAM,LGDISP,n ( $n > 0$ ) or with NLMOPTS,LRGSTRN,n ( $n > 0$ ), the strains are typically Logarithmic strains and the stresses are Cauchy Stresses.

## TABLE3D

## Tabular Function with Three Variables

Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.

Format:

| 1       | 2      | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|--------|------|----|----|----|----|----|----|----|
| TABLE3D | TID    | X0   | Y0 | Z0 | F0 |    |    |    |    |
|         | X1     | Y1   | Z1 | F1 | X2 | Y2 | Z2 | F2 |    |
|         | X3     | Y3   | Z3 | F3 | X4 | Y4 | Z4 | F4 |    |
|         | -etc.- | ENDT |    |    |    |    |    |    |    |

Example:

|         |      |     |     |       |      |      |      |       |  |
|---------|------|-----|-----|-------|------|------|------|-------|--|
| TABLE3D | 128  | 0.  | 0.  | 1.    |      |      |      |       |  |
|         | 7.   | 8.  | 9.  | 100.  | 12.  | 14.  | 11.  | 200.  |  |
|         | 17.  | 18. | 19. | 1100. | 112. | 114. | 111. | 1200. |  |
|         | ENDT |     |     |       |      |      |      |       |  |

| Descriptor | Meaning                              | Type        | Default  |
|------------|--------------------------------------|-------------|----------|
| TID        | Table identification number.         | Integer > 0 | Required |
| X0,Y0,Z0   | Offset of the independent variables. | Real        | 0.0      |
| F0         | Offset of the dependent variables.   | Real        | 0.0      |
| Xi,Yi,Zi   | Independent variables.               | Real        | 0.0      |
| Fi         | Dependent variable.                  | Real        | 0.0      |

Remarks:

- At least two continuation entries must be specified.
- The value of the function at  $(x,y,z)$  is calculated as

$$f = \frac{\sum_{i=1}^4 \frac{F_i - F_0}{d_i}}{\sum_{i=1}^4 \frac{1}{d_i}}$$

where  $f$  are the function values at the four points with the lowest value of

$$d_i^2 = (x - X0 - Xi)^2 + (y - Y0 - Yi)^2 + (z - Z0 - Zi)^2$$

**TABLED1**

## Dynamic Load Tabular Function, Form 1

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

**Format:**

| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9      | 10 |
|---------|-----|-------|-------|----|----|----|--------|--------|----|
| TABLED1 | TID | XAXIS | YAXIS |    |    |    |        |        |    |
|         | x1  | y1    | x2    | y2 | x3 | y3 | -etc.- | "ENDT" |    |

**Example:**

|         |      |     |     |     |     |     |      |  |  |
|---------|------|-----|-----|-----|-----|-----|------|--|--|
| TABLED1 | 32   |     |     |     |     |     |      |  |  |
|         | -3.0 | 6.9 | 2.0 | 5.6 | 3.0 | 5.6 | ENDT |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)   |
| XAXIS      | Specifies a linear or logarithmic interpolation for the x-axis. See Remarks 6. and 10.<br>(Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| YAXIS      | Specifies a linear or logarithmic interpolation for the y-axis. See Remarks 6. and 10.<br>(Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| xi, yi     | Tabular values. (Real)   |
| "ENDT"     | Flag indicating the end of the table.  |

**Remarks:**

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-160](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-160](#), the value of y at  $x = x_3$  is  $y = (y_3 + y_4)/2$ . If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as  $y = \sqrt{y_3 y_4}$ .
3. At least one continuation must be specified.
4. Any xi-yi pair may be ignored by placing the character string "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED1 uses the algorithm

$$y = y_T(x)$$

where  $x$  is input to the table and  $y$  is returned. The table look-up is performed using interpolation within the table and extrapolation outside the table using the two starting or end points. See [Figure 9-160](#). The algorithms used for interpolation or extrapolation are:

| XAXIS  | YAXIS  | $y_T(x)$   |
|--------|--------|--|
| LINEAR | LINEAR | $\frac{x_j - x}{x_j - x_i} y_i + \frac{x - x_i}{x_j - x_i} y_j$                                      |
| LOG    | LINEAR | $\frac{\ln(x_j/x)}{\ln(x_j/x_i)} y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} y_j$                          |
| LINEAR | LOG    | $\exp\left[\frac{x_j - x}{x_j - x_i} \ln y_i + \frac{x - x_i}{x_j - x_i} \ln y_j\right]$             |
| LOG    | LOG    | $\exp\left[\frac{\ln(x_j/x)}{\ln(x_j/x_i)} \ln y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} \ln y_j\right]$ |

where  $x_j$  and  $y_j$  follow  $x_i$  and  $y_i$ .

No warning messages are issued if table data is input incorrectly.

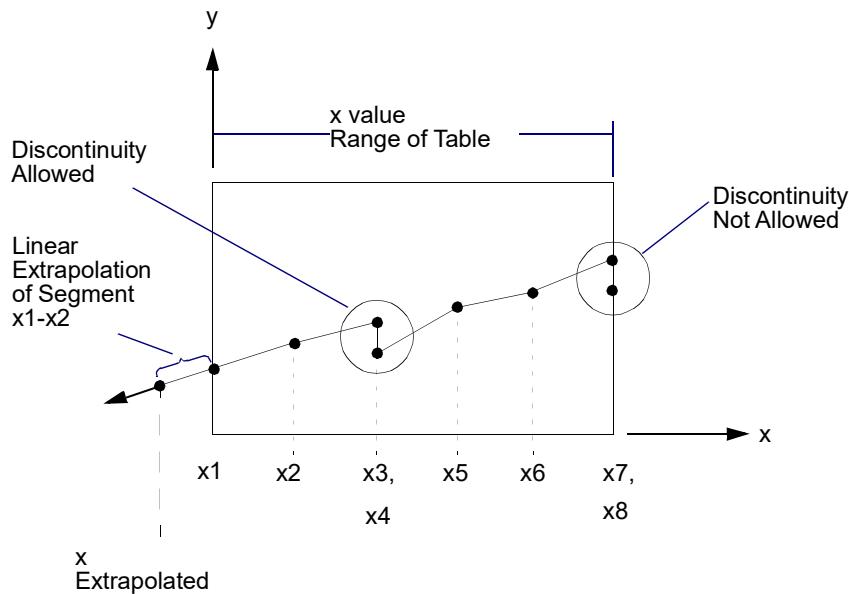


Figure 9-160 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads,  $x_i$  is measured in cycles per unit time.

9. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value  $\leq 0$ .
10. LOG is not supported for SOL 600 or SOL 700. Fields 3 and 4 must be blank.
11. RC network solver only supports LINEAR type XAXIS and YAXIS for thermal analysis.
12. The X-Y plot information resulting from a dynamic response solution (like any response versus frequency in SOLs 108/111 or any response versus time in SOLs 109/112) can be generated in TABLED1 format by using the XYTRAN module via DMAP. The 6th parameter of this module, which is an integer value, can be used for this purpose. Details can be obtained by referring to the description of this module in the DMAP Programmer's Guide.

## TABLED2

## Dynamic Load Tabular Function, Form 2

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLED2 | TID | X1 |    |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |      |       |     |      |      |     |     |     |  |
|---------|------|-------|-----|------|------|-----|-----|-----|--|
| TABLED2 | 15   | -10.5 |     |      |      |     |     |     |  |
|         | 1.0  | -4.5  | 2.0 | -4.2 | 2.0  | 2.8 | 7.0 | 6.5 |  |
|         | SKIP | SKIP  | 9.0 | 6.5  | ENDT |     |     |     |  |

| Descriptor | Meaning                                    |
|------------|--|
| TID        | Table identification number. (Integer > 0) |
| X1         | Table parameter. See Remark 6. (Real)      |
| xi, yi     | Tabular values. (Real)                     |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-161](#) discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-161](#), the value of y at x = x3 is  $y = (y_3 + y_4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED2 uses the algorithm

$$y = y_T(x - X1)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-161](#). No warning messages are issued if table data is input incorrectly.

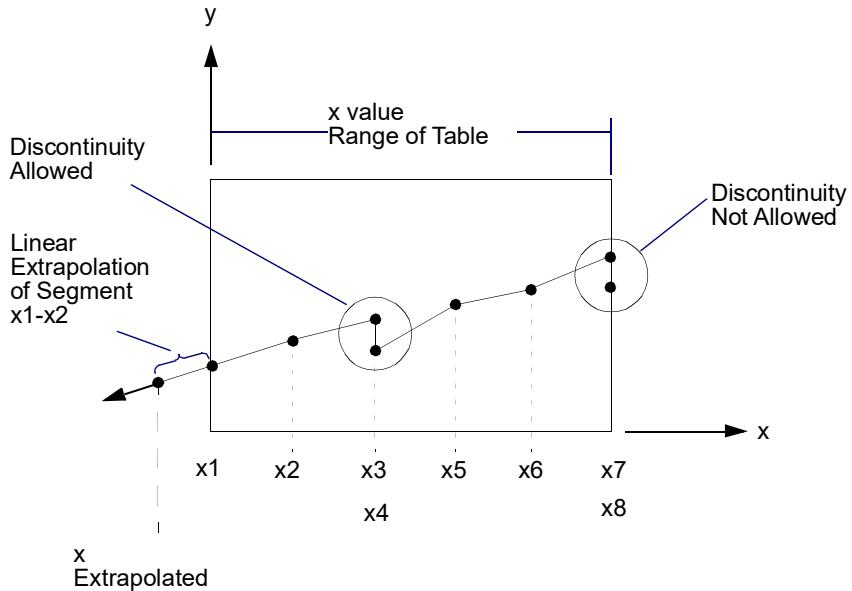


Figure 9-161 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads,  $x_1$  and  $x_i$  are measured in cycles per unit time.

## TABLED3

## Dynamic Load Tabular Function, Form 3

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLED3 | TID | X1 | X2 |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |     |       |      |     |     |     |      |  |  |
|---------|-----|-------|------|-----|-----|-----|------|--|--|
| TABLED3 | 62  | 126.9 | 30.0 |     |     |     |      |  |  |
|         | 2.9 | 2.9   | 3.6  | 4.7 | 5.2 | 5.7 | ENDT |  |  |

| Descriptor | Meaning                                    |
|------------|--|
| TID        | Table identification number. (Integer > 0) |
| X1, X2     | Table parameters. (Real; X2 ≠ 0.0)         |
| xi, yi     | Tabular values. (Real)                     |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-162](#) discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-162](#), the value of y at x = x3 is y = (y3 + y4)/2 .
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED3 uses the algorithm

$$y = y_T \left( \frac{x - X1}{X2} \right)$$

where x is input to the table and y is returned. The table look-up is performed using interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-162](#). No warning messages are issued if table data is input incorrectly.

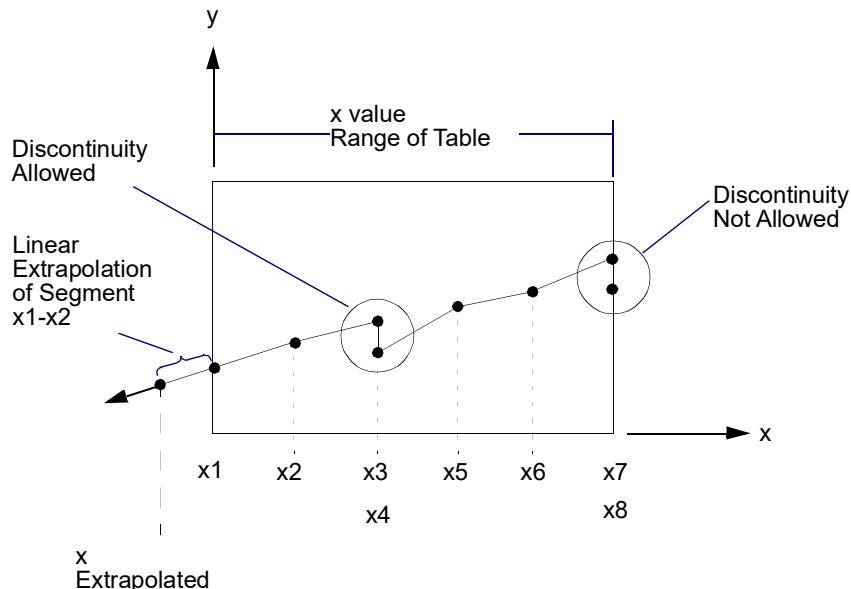


Figure 9-162 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, X1, X2, and xi are measured in cycles per unit time.

## TABLED4

## Dynamic Load Tabular Function, Form 4

Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLED4 | TID | X1 | X2 | X3 | X4 |    |        |   |    |
|         | A0  | A1 | A2 | A3 | A4 | A5 | -etc.- |   |    |

Example:

|         |      |         |        |     |        |      |  |  |  |
|---------|------|---------|--------|-----|--------|------|--|--|--|
| TABLED4 | 28   | 0.0     | 1.0    | 0.0 | 100.   |      |  |  |  |
|         | 2.91 | -0.0329 | 6.51-5 | 0.0 | -3.4-7 | ENDT |  |  |  |

| Descriptor | Meaning                                     |
|------------|---|
| TID        | Table identification number. (Integer > 0)  |
| Xi         | Table parameters. (Real; X2 ≠ 0.0; X3 < X4) |
| Ai         | Coefficients. (Real)                        |

Remarks:

- At least one continuation entry must be specified.
- The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLED4 uses the algorithm

$$y = \sum_{i=0}^N A_i \left( \frac{x - X_1}{X_2} \right)^i$$

where  $x$  is input to the table,  $y$  is returned, and  $N$  is the number of pairs. Whenever  $x < X_3$ , use  $X_3$  for  $x$ ; whenever  $x > X_4$ , use  $X_4$  for  $x$ . There are  $N + 1$  entries in the table. There are no error returns from this table look-up procedure.

- For frequency-dependent loads,  $xi$  is measured in cycles per unit time.

**TABLED5**

## Dynamic Load Tabular Function, Form 5

Defines a value as a function of two variables for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

| 1       | 2    | 3      | 4    | 5      | 6    | 7      | 8    | 9      | 10 |
|---------|------|--------|------|--------|------|--------|------|--------|----|
| TABLED5 | TID  |        |      |        |      |        |      |        |    |
|         | X(1) | TID(1) | X(2) | TID(2) | X(3) | TID(3) | X(4) | TID(4) |    |
|         | ...  | ...    | ENDT |        |      |        |      |        |    |

Example:

|         |     |     |      |     |      |     |      |  |  |
|---------|-----|-----|------|-----|------|-----|------|--|--|
| TABLED5 | 52  |     |      |     |      |     |      |  |  |
|         | 0.0 | 101 | 10.0 | 102 | 30.0 | 103 | ENDT |  |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. (Integer > 0)  |
| X(i)       | X value for the function specified by TID(i) (Real; no Default).  |
| TID(i)     | ID of a TABLED1, TABLED2, TABLED3 or TABLED4 defining the function Y for the given value of X. (Integer > 0; no Default). |

Remarks:

1. This table returns a value that is a function of 2 variables  $f(x, y)$ . The first variable value  $x$  is specified on this entry as X(i), the function versus  $y$  for the specified value for X(i) is specified on the referenced table TID(i).
2. When used with the NLRGAP entry to define a frequency dependent gap force as a function of penetration, the frequency values are input as X(i) and the variation of force vs. penetration is input on a series of TABLED1 entries referenced by the table values TID(i), one for each required change in properties with respect to frequency.
3. 2D linear interpolation is carried out for values of  $(x, y)$  not specified on the tabular entries.
4. X(i) values must be in ascending order.
5. TABLED5 Usage for frequency as function of temperature:

## TABLED5 usage

|      | ID | E      | G | NU |  |  |  | GE  |  |
|------|----|--------|---|----|--|--|--|-----|--|
| MAT1 | 33 | 7.2+10 |   | .3 |  |  |  | .02 |  |

|       |    |     |     |     |  |  |  |     |  |
|-------|----|-----|-----|-----|--|--|--|-----|--|
| MAT1F | 33 | 110 | 111 | 112 |  |  |  | 200 |  |
|-------|----|-----|-----|-----|--|--|--|-----|--|

|         |     |   |     |   |      |   |     |  |  |
|---------|-----|---|-----|---|------|---|-----|--|--|
| TABLED5 | 200 |   |     |   |      |   |     |  |  |
|         | 0.  | 3 | 40. | 4 | 100. | 5 | ... |  |  |

|         |   |     |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|
| TABLED1 | 3 | ... |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|

|         |   |     |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|
| TABLED1 | 4 | ... |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|

|         |   |     |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|
| TABLED1 | 5 | ... |  |  |  |  |  |  |  |
|---------|---|-----|--|--|--|--|--|--|--|

## Usage Example

E is dependent only on frequency, therefore points to a TABLED1 entry.

GE is both temperature and frequency dependent and therefore points to a TABLED5 entry

|         |     | E      | G      | NU     |      |        |        | GE   |  |
|---------|-----|--------|--------|--------|------|--------|--------|------|--|
| MAT1    | 1   | 7.2+10 | 2.8+10 | .3     |      |        | 2.22-5 | 0.02 |  |
| MAT1F   | 1   | 110    | 111    | 112    |      |        |        | 200  |  |
| TABLED1 | 110 |        |        |        |      |        |        |      |  |
|         | 10. | 7.2+10 | 200.   | 7.1+10 | 300. | 6.9+10 | ENDT   |      |  |
| TABLED1 | 111 |        |        |        |      |        |        |      |  |
|         | 10. | 2.8+10 | 200.   | 2.7+10 |      | 2.6+10 | ENDT   |      |  |
| TABLED1 | 112 |        |        |        |      |        |        |      |  |
|         | 10. | .3     | 200.   | .3     |      | .3     | ENDT   |      |  |

GE has frequency as a function of temperature

|         |     |      |      |   |      |       |      |  |  |
|---------|-----|------|------|---|------|-------|------|--|--|
| TABLED5 | 200 |      |      |   |      |       |      |  |  |
|         | 0.  | 3    | 40.  | 4 | 100. | 5     | ENDT |  |  |
| TABLED1 | 3   |      |      |   |      |       |      |  |  |
|         | 10. | 0.02 | 200. |   |      | 0.025 | ENDT |  |  |

|         |     |       |      |      |      |       |      |  |
|---------|-----|-------|------|------|------|-------|------|--|
| TABLED1 | 4   |       |      |      |      |       |      |  |
|         | 10. | 0.025 | 200. |      |      | 0.03  | ENDT |  |
| TABLED1 | 5   |       |      |      |      |       |      |  |
|         | 10. | 0.03  | 200. | 0.04 | 300. | 0.035 | ENDT |  |

TABLED1 units are x=frequency, y=material value

TABLED5 input is numerical temperature - frequency table ID

For an element with average temperature of 15.0 degrees the GE value will be selected from TABLED1 ID=3;

For an element with average temperature of 30.0 degrees the GE value will be selected from TABLED1 ID=4;

For an element with average temperature of 20.0 degrees the GE value will be selected from TABLED1 ID=3;

6. TABLED5 in Solution Sequence SOL108, SOL111, SO200, or SOL400 with ANALYSIS=DFREQ or MFREQ:

Example (1): TABLED5 specification using TEMP(INIT) or TEMP(MATE) entry, no TEMP(LOAD) in Case Control section.

```
SOL 108
...
TEMP (INIT) =5
BEGIN BULK
...
TEMP, 5, ...
...
ENDDATA
```

The temperatures specified by TEMP(INIT) will be used for TABLED5 look up

Example (2): TABLED5 specification using TEMP(LOAD) entry.

```
SOL 108
...
TEMP (INIT) =5
SUBCASE 1
TEMP (LOAD) =12
...
```

```
BEGIN BULK
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up.

User is reminded that for dynamic solutions sequences, the above will not apply thermal loading unless there is a DLOAD entry pointing to an RLOAD1 or RLOAD2 referencing TEMP(LOAD) with ID=12 specified.

Example (3): TABLED5 specification using TEMP(LOAD) entry and thermal loading desired.

```
SOL 108
...
TEMP (INIT)=5
SUBCASE 1
TEMP (LOAD)=12
DLOAD = 50
...
BEGIN BULK
...
RLOAD, 50, 12, , , 1.0, , LOAD
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up and will also be used to compute thermal loads in elements.

**TABLEHT****Heat Transfer Coefficient Table with Two Variables**

Specifies a function of two variables for convection heat transfer coefficient.

**Format:**

| 1       | 2   | 3    | 4  | 5    | 6  | 7     | 8 | 9 | 10 |
|---------|-----|------|----|------|----|-------|---|---|----|
| TABLEHT | TID |      |    |      |    |       |   |   |    |
|         | x1  | TID1 | x2 | TID2 | x3 | -etc. |   |   |    |

**Example:**

|         |      |     |      |     |      |     |      |  |  |
|---------|------|-----|------|-----|------|-----|------|--|--|
| TABLEHT | 85   |     |      |     |      |     |      |  |  |
|         | 10.0 | 101 | 25.0 | 102 | 40.0 | 110 | ENDT |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)                     |
| xi         | Independent variables. (Real)                                  |
| TIDI       | Table identification numbers of TABLEH1 entries. (Integer > 0) |

**Remarks:**

1. xi must be listed in ascending order.
2. At least one continuation entry must be present.
3. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by PCONV entries that define free convection boundary condition properties.

## TABLEH1

## Heat Transfer Coefficient Table, Form 1

Defines a tabular function referenced by TABLEHT for convection heat transfer coefficient.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7      | 8 | 9 | 10 |
|---------|-----|----|----|----|----|--------|---|---|----|
| TABLEH1 | TID |    |    |    |    |        |   |   |    |
|         | y1  | f1 | y2 | f2 | y3 | -etc.- |   |   |    |

Example:

|         |      |      |      |      |       |      |      |  |  |
|---------|------|------|------|------|-------|------|------|--|--|
| TABLEH1 | 123  |      |      |      |       |      |      |  |  |
|         | 50.0 | 5.23 | 75.0 | 3.76 | 110.0 | 0.97 | ENDT |  |  |

| Descriptor | Meaning                                    |
|------------|--|
| TID        | Table identification number. (Integer > 0) |
| yi         | Independent variables. (Real)              |
| fi         | Dependent variable. (Real)                 |

Remarks:

1. yi must be listed in ascending order.
2. At least one continuation entry must be present.
3. Any yi-fi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
5. TABLEH1 is used to input a curve in the form of  

$$f = f(y)$$
 where  $y$  is input to the table and  $f$  is returned. The table look-up is performed using linear interpolation within the table and is evaluated at the starting or end point outside the table. No warning messages are issued if table data is input incorrectly.
6. Discontinuities are not recommended and may lead to unstable results.

**TABLE1**

Specifies a Table of Amplitude vs Pseudo-Time for Static Loads Specified in LDTABL Entries

Specifies a table of amplitude vs pseudo-time for static loads specified in LDTABL entries SOL 600 only.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8    | 9      | 10 |
|---------|-----|----|----|----|----|----|------|--------|----|
| TABLEL1 | TID |    |    |    |    |    |      |        |    |
|         | X1  | Y1 | X2 | Y2 | X3 | Y3 | etc. | "ENDT" |    |

Example:

|         |     |     |     |     |     |     |      |  |  |
|---------|-----|-----|-----|-----|-----|-----|------|--|--|
| TABLEL1 | 101 |     |     |     |     |     |      |  |  |
|         | 0.0 | 0.0 | 0.5 | 1.0 | 1.0 | 0.0 | ENDT |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0; no Default) |
| Xi, Yi     | Tabular values. (Real; no Default)                     |
| "ENDT"     | Flag indicating end of the table. (Character)          |

Remarks:

1. TABLEL1 can only be used in SOL 600.
2. TABLED1 may be used instead of TABLEL1, however, XAXIS and YAXIS must be LINEAR or blank.
3. The TID must be unique among all TABLEL1's and TABLED1's.
4. The string "SKIP" used in TABLED1 may not be used in TABLEL1.
5. Log X and Y is not available in TABLEL1.
6. No blank fields are allowed starting with the second field of the first continuation line until the ENDT string.
7. There must be at least two points in the table.

## TABLEM1

## Material Property Table, Form 1

Defines a tabular function for use in generating temperature-dependent material or fatigue related properties.

Format:

| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9      | 10 |
|---------|-----|-------|-------|----|----|----|--------|--------|----|
| TABLEM1 | TID | XAXIS | YAXIS |    |    |    |        |        |    |
|         | x1  | y1    | x2    | y2 | x3 | y3 | -etc.- | "ENDT" |    |

Example:

|         |      |     |     |     |     |     |      |  |  |
|---------|------|-----|-----|-----|-----|-----|------|--|--|
| TABLEM1 | 32   |     |     |     |     |     |      |  |  |
|         | -3.0 | 6.9 | 2.0 | 5.6 | 3.0 | 5.6 | ENDT |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. See Remark 8. (Integer > 0 or Integer < 0)  |
| XAXIS      | Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") See Remark 9. |
| YAXIS      | Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") See Remark 9. |
| xi, yi     | Tabular values. (Real)   |
| "ENDT"     | Flag indicating the end of the table.  |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-163](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-163](#), the value of y at x = x3 is  $y = (y_3 + y_4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM1 uses the algorithm

$$y = y_T(x) \quad (\text{for heat transfer, see Remark 7.})$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-163](#). No warning messages are issued if table data is input incorrectly.

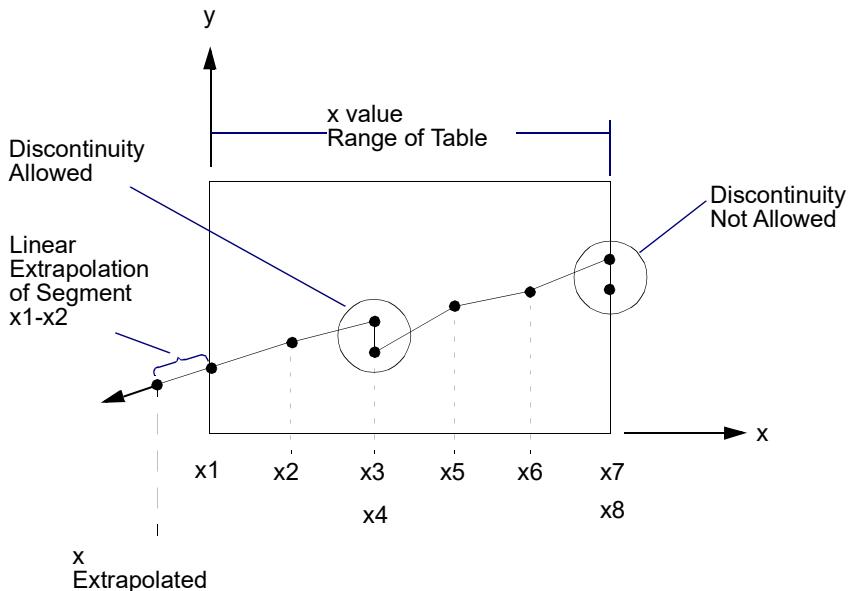


Figure 9-163 Example of Table Extrapolation and Discontinuity

- For Nastran heat transfer, the TABLEM1 assumes

$$y = zy_T(x)$$

where  $x$  is input to the table,  $y$  is returned and  $z$  is supplied from MAT4 or MT5 entries.

- A negative TID is used to associate thermal strain  $\epsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 10000000$ .
- Logarithmic XAXIS and YAXIS is only recognized when specifying S-N curves referenced by MATFTG entries for TYPE=TABLE. All other uses use linear and ignore these fields.
- TABLEM1 in RESTART job will cause recalculation of eigenvalues even in the cases where TABLEM1 does not change stiffness and/or mass matrix. An example is SOL 111 RESTART for fatigue analysis with TABLEM1 referenced on MATFTG.

## TABLEM2

## Material Property Table, Form 2

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLEM2 | TID | X1 |    |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |      |       |     |      |      |     |     |     |  |
|---------|------|-------|-----|------|------|-----|-----|-----|--|
| TABLEM2 | 15   | -10.5 |     |      |      |     |     |     |  |
|         | 1.0  | -4.5  | 2.0 | -4.5 | 2.0  | 2.8 | 7.0 | 6.5 |  |
|         | SKIP | SKIP  | 9.0 | 6.5  | ENDT |     |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. See Remark 7. (Integer > 0 or Integer < 0) |
| X1         | Table parameter. (Real)   |
| xi, yi     | Tabular values. (Real)  |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-164](#), discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-164](#), the value of y at x = x3 is  $y = (y3 + y4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM2 uses the algorithm

$$y = zy_T(x - X1)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-164](#). No warning messages are issued if table data is input incorrectly.

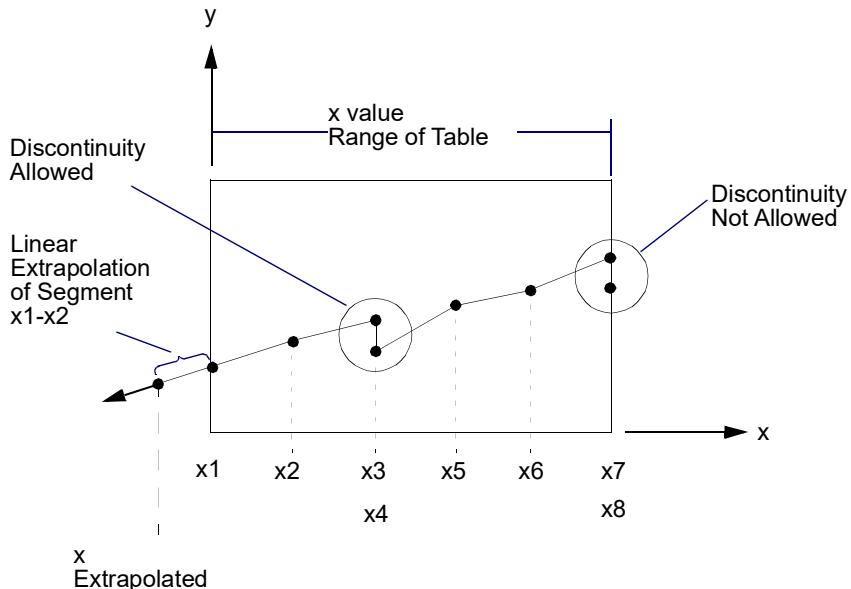


Figure 9-164 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain  $\varepsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 100000000$ .

## TABLEM3

## Material Property Table, Form 3

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLEM3 | TID | X1 | X2 |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |     |       |      |     |     |     |      |  |  |
|---------|-----|-------|------|-----|-----|-----|------|--|--|
| TABLEM3 | 62  | 126.9 | 30.0 |     |     |     |      |  |  |
|         | 2.9 | 2.9   | 3.6  | 4.7 | 5.2 | 5.7 | ENDT |  |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. See Remark 7. (Integer > 0 or Integer < 0) |
| X1, X2     | Table parameters. See Remark 6. (Real; X2 ≠ 0.0)                        |
| xi, yi     | Tabular values. (Real)  |

Remarks:

1. Tabular values for xi must be specified in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-165](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-165](#), the value of y at x = x3 is  $y = (y3 + y4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM3 uses the algorithm

$$y = zy_T \left( \frac{x - X1}{X2} \right)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-165](#). No warning messages are issued if table data is input incorrectly.

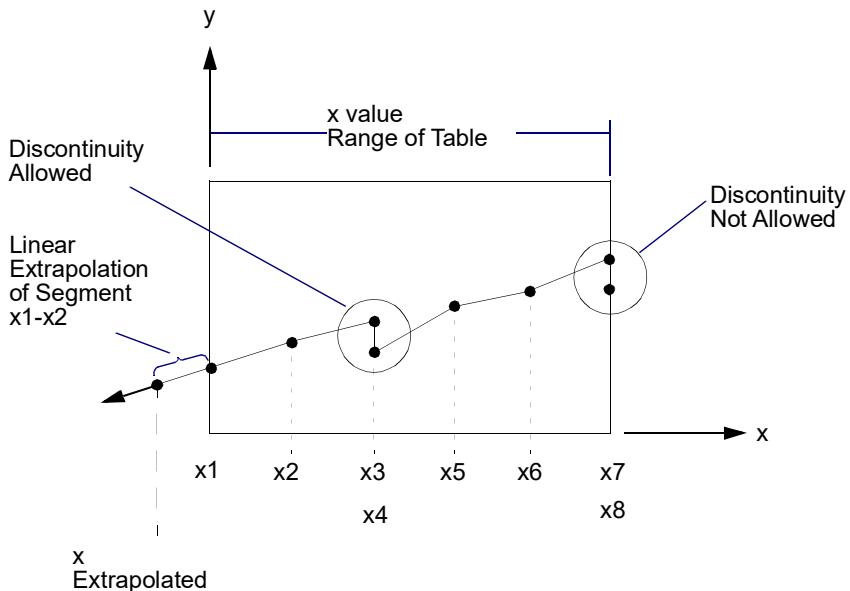


Figure 9-165 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain  $\varepsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 100000000$ .

## TABLEM4

## Material Property Table, Form 4

Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLEM4 | TID | X1 | X2 | X3 | X4 |    |        |   |    |
|         | A0  | A1 | A2 | A3 | A4 | A5 | -etc.- |   |    |

Example:

|         |      |         |        |     |        |      |  |  |  |
|---------|------|---------|--------|-----|--------|------|--|--|--|
| TABLEM4 | 28   | 0.0     | 1.0    | 0.0 | 100.   |      |  |  |  |
|         | 2.91 | -0.0329 | 6.51-5 | 0.0 | -3.4-7 | ENDT |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. See Remark 4. (Integer > 0 or Integer < 0) |
| Xi         | Table parameters. (Real; X2 ≠ 0.0; X3 < X4)                             |
| Ai         | Coefficients. (Real)  |

Remarks:

- At least one continuation entry must be specified.
- The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLEM4 uses the algorithm

$$y = z \sum_{i=0}^N A_i \left( \frac{x - X_1}{X_2} \right)^i$$

where  $x$  is input to the table,  $y$  is returned and  $z$  is supplied from the MATi entry. Whenever  $x < X_3$ , use  $X_3$  for  $x$ ; whenever  $x > X_4$ , use  $X_4$  for  $x$ . There are  $N + 1$  entries in the table. There are no error returns from this table look-up procedure.

- A negative TID is used to associate thermal strain  $\varepsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 100000000$ .

**TABLES1****Material Property Table, Form 1**

Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

**Format:**

| 1       | 2   | 3    | 4  | 5  | 6  | 7  | 8      | 9      | 10 |
|---------|-----|------|----|----|----|----|--------|--------|----|
| TABLES1 | TID | TYPE |    |    |    |    |        |        |    |
|         | x1  | y1   | x2 | y2 | x3 | y3 | -etc.- | "ENDT" |    |

**Example:**

|         |     |     |     |        |     |        |      |  |  |
|---------|-----|-----|-----|--------|-----|--------|------|--|--|
| TABLES1 | 32  |     |     |        |     |        |      |  |  |
|         | 0.0 | 0.0 | .01 | 10000. | .02 | 15000. | ENDT |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)   |
| TYPE       | Flag to define type of the stress-strain curve. See Remark 10. (Integer = 1 or 2; Default = 1) |
| xi, yi     | Tabular values. (Real)   |
| "ENDT"     | Flag indicating the end of the table.  |

**Remarks:**

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-166](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-166](#), the value of y at x = x3 is  $y = (y_3 + y_4)/2$ .
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
6. TABLES1 is used to input a curve in the form of

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-166](#). No warning messages are issued if table data is input incorrectly.

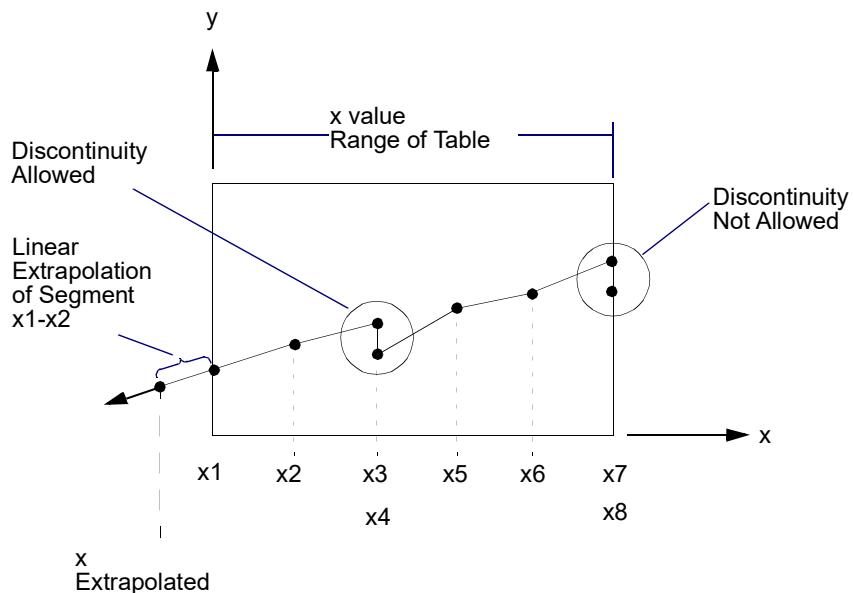


Figure 9-166 Example of Table Extrapolation and Discontinuity

7. Discontinuities are not recommended and may lead to unstable results. Discontinuities are not allowed in nonlinear solution sequences.
8. For SOL 600, general temperature-dependent stress vs. plastic strain curves may be entered using a combination of TABLEST and TABLES1 entries. Each TABLES1 entry is at a constant temperature. All entries must be in the form of stress vs. plastic strain using the stress and strain measures to be incorporated into the analysis. All sets of stress-strain values for a particular TABLES1 entry must be at the same temperature. One set is required for the lowest temperature in the model and another at or above the highest temperature in the model.
9. For SOL 600, the stress and strain values entered here depend on the stress and strain measures selected for the analysis. In addition, the strain is controlled using PARAM,MRTABLS1 which provides several methods of converting an engineering stress-strain curve to a stress vs. plastic strain curve (see MRTABLS1 in the Parameters Section).
10. For SOL 400, TYPE denotes the type of stress-strain curve; 1 - Cauchy (true) stress vs. total true strain; and 2 - Cauchy (true) stress vs. plastic true strain. For MATS1 Bulk Data entry, only TYPE = 1 can be used. A user fatal error will be issued if TYPE = 2 is used. For MATEP Bulk Data entry both TYPE = 1 and 2 can be used.

**TABLEST****Material Property Temperature-Dependence Table**

Specifies the material property tables for nonlinear elastic temperature-dependent materials.

**Format:**

| 1       | 2   | 3    | 4  | 5    | 6  | 7      | 8 | 9 | 10 |
|---------|-----|------|----|------|----|--------|---|---|----|
| TABLEST | TID |      |    |      |    |        |   |   |    |
|         | T1  | TID1 | T2 | TID2 | T3 | -etc.- |   |   |    |

**Example:**

|         |       |    |       |    |      |  |  |  |  |
|---------|-------|----|-------|----|------|--|--|--|--|
| TABLEST | 101   |    |       |    |      |  |  |  |  |
|         | 150.0 | 10 | 175.0 | 20 | ENDT |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)                     |
| Ti         | Temperature values. (Real)                                     |
| TIDi       | Table identification numbers of TABLES1 entries. (Integer > 0) |

**Remarks:**

1. TIDi must be unique with respect to all TABLES1 and TABLEST table identification numbers.
2. Temperature values must be listed in ascending order.
3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by MATS1 entries that define nonlinear elastic (TYPE = "NLELAST") materials.
5. For SOL 600, this entry provides IDs of TABLES1 curves as a function of temperature for use with Marc's AF\_flowmat. The strains are plastic strain for all curves entered. The first curve must be entered at the lowest temperature encountered in the analysis run. Curves must be defined that equal or exceed the maximum temperature encountered in the run.

**TABLFTG****Fatigue Loading Tabular Data**

Defines tabular data for specifying fatigue cyclic loading variation.

**Format:**

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9      | 10 |
|---------|-----|----|----|----|----|----|----|--------|----|
| TABLFTG | TID |    |    |    |    |    |    |        |    |
|         | y1  | y2 | y3 | y4 | y5 | y6 | y7 | "ENDT" |    |

**Example:**

|         |       |      |     |     |      |  |  |  |  |
|---------|-------|------|-----|-----|------|--|--|--|--|
| TABLFTG | 1     |      |     |     |      |  |  |  |  |
|         | 0.000 | -1.0 | 1.0 | 0.0 | ENDT |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)               |
| yi         | Y value of each point in the time history curve. (Real). |
| "ENDT"     | Flag indicating the end of the table.                    |

**Remarks:**

1. The TABLFTG is referenced by a FTGLOAD entry.
2. The x-values are assumed to be in ascending order. For rainflow cycle counting purposes the actual x values are inconsequential.
3. For modal analysis using SOL 103, this would define the modal participation factors for a particular mode.

**TABLRPC****Dynamic Load Tabular Function Referencing Channel Data File**

Defines a tabular function for use in generating time-dependent loads from an externally defined channel data file.

**Format:**

| 1       | 2                | 3                | 4                | 5                | 6                | 7                | 8      | 9      | 10 |
|---------|------------------|------------------|------------------|------------------|------------------|------------------|--------|--------|----|
| TABLRPC | TID              | XAXIS            | YAXIS            | UID              | TYPE             | CHAN             | TOTIM  | PUNCH  |    |
|         | x <sub>i</sub> 1 | x <sub>j</sub> 1 | x <sub>i</sub> 2 | x <sub>j</sub> 2 | x <sub>i</sub> 3 | x <sub>j</sub> 3 | -etc.- | "ENDT" |    |

**Examples:**

|         |                  |  |  |    |     |   |  |  |  |
|---------|------------------|--|--|----|-----|---|--|--|--|
| TABLRPC | 32               |  |  | 33 | RPC | 5 |  |  |  |
| UDNAME  | 33               |  |  |    |     |   |  |  |  |
|         | C:\myrpcfile.rsp |  |  |    |     |   |  |  |  |

|         |                  |      |      |      |      |  |  |  |  |
|---------|------------------|------|------|------|------|--|--|--|--|
| TABLRPC | 32               |      |      | 33   | DAC  |  |  |  |  |
|         | 0.0              | 20.0 | 35.0 | 60.0 | ENDT |  |  |  |  |
| UDNAME  | 33               |      |      |      |      |  |  |  |  |
|         | C:\myrpcfile.dac |      |      |      |      |  |  |  |  |

|         |                  |      |      |      |      |   |  |  |  |
|---------|------------------|------|------|------|------|---|--|--|--|
| TABLRPC | 32               |      |      | -33  |      | 5 |  |  |  |
|         | 0.0              | 20.0 | 35.0 | 60.0 | ENDT |   |  |  |  |
| UDNAME  | 33               |      |      |      |      |   |  |  |  |
|         | C:\myrpcfile.rsp |      |      |      |      |   |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| TID        | Table identification number. (Integer > 0, no default)  |
| XAXIS      | Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")  |
| YAXIS      | Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")  |
| UID        | Identification number of a UDNAME entry to specify the external file that defines the x-y pair values that define the actual tabular function. Required. (Integer != 0, no default) |
| TYPE       | Type of external file to specify via UID field. (Character: "RPC", "DAC", or "S3T"; Default = "RPC")  |
| CHAN       | Channel number to read for Y (load) values from "RPC" and "S3T" files. Leave blank for "DAC" files. (Integer > 0, Default = 1)  |
| TOTIM      | Total time of the signal defined in channel CHAN. Optional. (Real > 0., Default=Blank). See remarks below.  |

| Descriptor    | Meaning   |
|---------------|---|
| PUNCH         | Specify whether equivalent TABLED1 entries containing the actual x-y pairs should be written to the PUNCH file. (Character: "YES" or "NO"; Default= "NO")   |
| $x_i^1/x_j^1$ | x-y pair Filter mechanism. The given $(x_i^1, x_j^1)$ pairs reflect the start and end x-values for inclusion from the external file; or if UID is negative, the given pairs reflect the start and end x-values to exclude from reading from the external file. See remarks below. Optional. (Real). |
| "ENDT"        | Flag indicating the end of the table.   |

#### Remarks:

1. The TABLRPC entry can be referenced by any entry that can reference a TABLED1. The TABLRPC is internally converted to a TABLED1 entry using the channel data specified. This internal TABLED1 entry can be written to the punch (.pch) file if the PUNCH field is set to YES.
2. The CHANnel specified contains the Y (load) data. The X (time) data for each point is automatically extracted from the channel file at each point to create the x-y pairs for a TABLED1 entry. The TOTIM (total time) of the signal can be optionally supplied, in which case the time increment of each point is calculated as TOTIM / (NPNTS-1) where NPNTS are the number of signal points. This will override any time specifications for the points from the original channel file.
3.  $x_i^1/x_j^1$  field pairs are purely optional. If not needed, no continuation lines should be present, in which case the entire signal defined in the specified CHANnel is taken.
4. The  $x_i^1/x_j^1$  field pairs represent sections of the channel to read and must be in the order in which they appear in the external file. For example, to read from only  $x=0.0$  to  $x=1.0$ , specify  $x_1^1=0.0$  and  $x_1^2=1.0$  and  $x_2^1=ENDT$ . If the first  $x_i^1$  is left blank, reading begins at the first data point. If the last  $x_i^1$  value is left blank, then the rest of the data points are read to the end of the channel. If a specified  $x_i^1$  or  $x_i^2$  value does not exist, only the points in between are retained or excluded.
5. Any  $x_i^1/x_j^1$  field pair may be ignored by placing the character string "SKIP" in either of the two fields.
6. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT"
7. As this data is converted to a TABLED1 entry, all comments pertaining to the TABLED1 entry are also valid.

**TABLUDS**

User Defined Failure For Nonlinear Explicit Analysis.

Specifies that a user routine is being used to define an arbitrary function. Use in SOL700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|---|---|---|---|---|----|
| TABLUDS | TID | GROUP | UNAME |   |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myfunc 'SCA.MDSolver.Obj.Uds.Dytran.Loads'
```

*In Bulk Data:*

| 1       | 2  | 3      | 4      | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|---|---|---|---|---|----|
| TABLUDS | 12 | myfunc | EXFUNC |   |   |   |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| TID        | Unique output number. (Integer > 0; Required)   |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement.<br>(Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXFUNC)                   |

**Remarks:**

1. Since tables and user-defined functions belong to the same group, the table numbers must be unique.
2. UNAME can be:

| Subroutine Name | Function                       |
|-----------------|--------------------------------|
| EXFUNC          | Standard user defined function |

**TABRND1****Power Spectral Density Table**

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

**Format:**

| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|-------|-------|----|----|----|--------|---|----|
| TABRND1 | TID | XAXIS | YAXIS |    |    |    |        |   |    |
|         | f1  | g1    | f2    | g2 | f3 | g3 | -etc.- |   |    |

**Example:**

|         |     |        |     |        |      |  |  |  |  |
|---------|-----|--------|-----|--------|------|--|--|--|--|
| TABRND1 | 3   |        |     |        |      |  |  |  |  |
|         | 2.5 | .01057 | 2.6 | .01362 | ENDT |  |  |  |  |

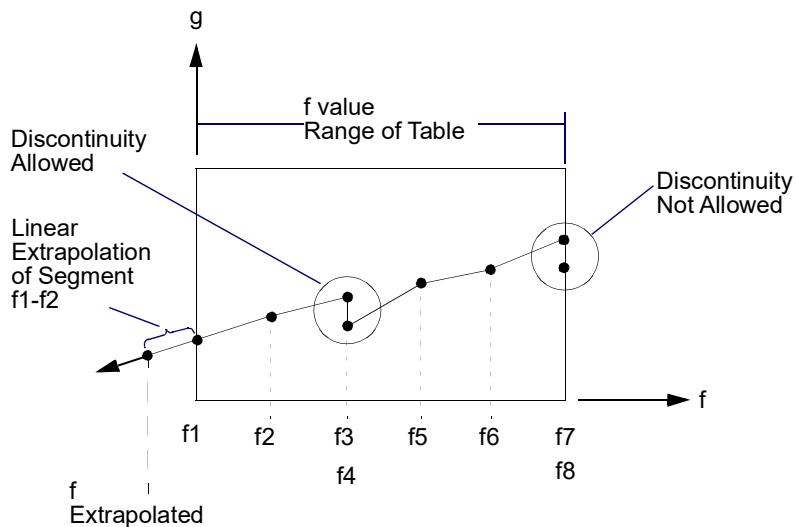
| Descriptor     | Meaning  |
|----------------|--|
| TID            | Table identification number. (Integer > 0)   |
| XAXIS          | Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| YAXIS          | Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| f <sub>i</sub> | Frequency value in cycles per unit time. (Real $\geq 0.0$ )  |
| g <sub>i</sub> | Power spectral density. (Real)   |

**Remarks:**

1. The f<sub>i</sub> must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-167](#) discontinuities are allowed only between points f<sub>2</sub> through f<sub>7</sub>. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-167](#), the value of g at f = f<sub>3</sub> is  $g = (g_3 + g_4)/2$ . If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as  $y = \sqrt{y_3 y_4}$ .
3. At least two entries must be present.
4. Any f<sub>i</sub>-g<sub>i</sub> pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABRND1 uses the algorithm

$$g = g_T(f)$$

where  $f$  is input to the table and  $g$  is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-167](#). No warning messages are issued if table data is input incorrectly.



[Figure 9-167](#) Example of Table Extrapolation and Discontinuity

7. For auto spectral density, the value of  $g$  returned must be greater than or equal to zero, as shown in [Remark 6](#).
8. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value  $\leq 0$ .
9. The algorithms used are:

| XAXIS  | YAXIS  | $f(x)$   |
|--------|--------|--|
| LINEAR | LINEAR | $\frac{f_{i+1}-f}{f_{i+1}-f_i}g_i + \frac{f-f_i}{f_{i+1}-f_i}g_{i+1}$  |
| LOG    | LINEAR | $\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}g_{i+1}$                          |
| LINEAR | LOG    | $\exp\left[\frac{f_{i+1}-f}{f_{i+1}-f_i}\ln g_i + \frac{f-f_i}{f_{i+1}-f_i}\ln g_{i+1}\right]$                     |
| LOG    | LOG    | $\exp\left[\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}\ln g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}\ln g_{i+1}\right]$ |

where  $f_i < f < f_{i+1}$ .

10. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TABRNDG****Gust Power Spectral Density**

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.

**Format:**

| 1       | 2   | 3    | 4   | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|-----|------|-----|----|---|---|---|---|----|
| TABRNDG | TID | TYPE | L/U | WG |   |   |   |   |    |

**Example:**

|         |      |   |     |    |  |  |  |  |  |
|---------|------|---|-----|----|--|--|--|--|--|
| TABRNDG | 1020 | 1 | 1.3 | .1 |  |  |  |  |  |
|---------|------|---|-----|----|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| TID        | Table identification number. (Integer > 0)   |
| TYPE       | PSD type: von Karman (TYPE = 1) or Dryden model (TYPE = 2). (Integer = 1 or 2)       |
| L/U        | Scale of turbulence divided by velocity (units of time). See L/U in Remark 2. (Real) |
| WG         | Root-mean-square gust velocity. (Real)   |

**Remarks:**

1. This entry must be referenced by a RANDPS entry.
2. The power spectral density is given by

$$S_q(\omega) = 2(WG)^2(L/U) \frac{1 + 2(p+1)k^2(L/U)^2\omega^2}{[1 + k^2(L/U)^2\omega^2]^{p+3/2}}$$

where:

| Type         | p   | k     |
|--------------|-----|-------|
| 1=von Karman | 1/3 | 1.339 |
| 2=Dryden     | 1/2 | 1.0   |

and  $\omega = 2\pi f$ . The units of  $S_q(\omega)$  are velocity squared per frequency ( $f$ ).

3. Other power spectral density functions may be defined using the TABRND1 entry.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TABSCTL****Load Stepping Control Table - SOL 400**

This option allows the user to provide user criteria for load stepping control. It is referred to by an NLSTEP entry. The criteria defined herein are used for controlling the load step size. The criteria come in two flavors as defined in the NLSTEP entry: limit and target. When used as limits, the time step will be reduced if the criterion would be violated. When used as target, the time step will also be increased if the calculated results are less than what is specified by the criteria.

The criteria are calculated for elements or grids as defined below. By default all elements or nodes are used for evaluating the respective criterion, but this can be limited to specific sets. (See the [NLSTEP, 2616](#) entry.)

**Format:**

| 1       | 2       | 3        | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|---------|----------|-------|-------|-------|-------|-------|-------|----|
| TABSCTL | TID     |          |       |       |       |       |       |       |    |
|         | ICRIT_1 | SET3_ID1 | YT1_1 | MT1_1 | YT2_1 | MT2_1 | YT3_1 | MT3_1 |    |
|         | YT4_1   | MT4_1    |       |       |       |       |       |       |    |
|         | ICRIT_2 | SET3_ID2 | YT1_2 | MT_2  | YT2_2 | MT2_1 | YT3_2 | MT3_2 |    |
|         | YT4_2   | MT4_2    |       |       |       |       |       |       |    |
|         | ...     | ...      | ...   | ...   |       |       |       |       |    |
|         | ICRIT_n | SET3_IDn | YT1_n | MT1_n | YT1_n | MT2_n | YT3_n | MT3_n |    |
|         | YT4_n   | MT4_n    |       |       |       |       |       |       |    |

**Example:**

|         |      |     |     |     |     |     |     |  |  |
|---------|------|-----|-----|-----|-----|-----|-----|--|--|
| TABSCTL | 17   |     |     |     |     |     |     |  |  |
|         | ESI  | 84  | .03 | 21. | 0.4 | 55. | 0.6 |  |  |
|         | ESRI | 18  | .03 | 21. |     |     |     |  |  |
|         | EPSI | ALL | 1.1 |     |     |     |     |  |  |

| Descriptor | Meaning  |
|------------|--|
| TID        | ID of TABSCTL entry. (Integer > 0)   |
| ICRITi     | Type of user criterion to use and the type of entity it refers to: (Character; no Default) |
| ESI        | Element Strain Increment   |
| EPSI       | Element Plastic Strain Increment   |
| ECSI       | Element Creep Strain Increment   |
| ENCSI      | Element Normalized Creep Strain Increment  |
| ESTRI      | Element STRes Increment  |
| ESRI       | Element Strain Energy Increment  |
| GTI        | Grid Temperature Increment   |

| Descriptor | Meaning   |
|------------|---|
| GDI        | Grid Displacement Increment   |
| GRI        | Grid Rotation Increment   |
| ENSTRI     | Element Normalized STRes Increment  |
| SET3_IDi   | ID of a SET3 Bulk Data entry. (Integer > 0 or (ALL or BLANK); Default = ALL)                              |
| YT1_i      | First limit or target value for the current criterion i. (Real; no Default)                               |
| MT1_i      | First range or target value for the current criterion i. See Remark 2. (Real-default blank-always active) |
| YT2_i      | Second limit or target value for criterion. (Real; no Default)  |
| MT2_i      | Second range of applicability for criterion i. See Remark 2. (Real-default blank-always active)           |

#### Remarks:

1. This entry is selected by the CRITTID field of the NLSTEP Bulk Data entry.
2. For each criterion ICRTT\_n, up to four ranges of target/limit values can be given. This allows the use of different limit/target values for different ranges of the corresponding total quality. For example, for the first criterion type, YT1\_1 is the strain increment while MT1\_1 is the largest total strain for which YT1\_1 will be used. Typically, the last MTn used should be zero in which case the corresponding YTn will be used for all larger total values.
3. The TID must be unique among all TABSCTL entries.

**TEMP****Grid Point Temperature Field**

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.

**Format:**

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| TEMP | SID | G1 | T1 | G2 | T2 | G3 | T3 |   |    |

**Example:**

|      |   |    |       |    |       |  |  |  |  |
|------|---|----|-------|----|-------|--|--|--|--|
| TEMP | 3 | 94 | 316.2 | 49 | 219.8 |  |  |  |  |
|------|---|----|-------|----|-------|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Temperature set identification number. (Integer > 0) |
| Gi         | Grid point identification number. (Integer > 0)      |
| Ti         | Temperature. (Real)                                  |

**Remarks:**

1. In the static solution sequences, SID must be selected by the TEMP Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to three grid point temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined. Gauss point temperatures are averaged for solid elements instead of grid point temperature.
7. For steady state heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC = SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.

**TEMPAX****Conical Shell Temperature**

Defines temperature sets for conical shell problems.

**Format:**

| 1      | 2    | 3    | 4    | 5  | 6    | 7    | 8    | 9  | 10 |
|--------|------|------|------|----|------|------|------|----|----|
| TEMPAX | SID1 | RID1 | PHI1 | T1 | SID2 | RID2 | PHI2 | T2 |    |

**Example:**

|        |   |   |      |       |  |  |  |  |  |
|--------|---|---|------|-------|--|--|--|--|--|
| TEMPAX | 4 | 7 | 30.0 | 105.3 |  |  |  |  |  |
|--------|---|---|------|-------|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SIDi       | Temperature set identification number. (Integer > 0)         |
| RIDI       | Ring identification number (see RINGAX entry). (Integer > 0) |
| PHIi       | Azimuthal angle in degrees. (Real)                           |
| Ti         | Temperature. (Real)  |

**Remarks:**

1. TEMPAX is allowed only if an AXIC entry is also present.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. Temperature sets must be selected with the Case Control command TEMP=SID.
4. One or two temperatures may be defined on each entry.
5. For a discussion of the conical shell problem, see [Restart Procedures](#) in the *MSC Nastran Reference Guide*.
6. TEMP(INIT) is not used with this entry.

**TEMPB3****CBEAM3 Element Temperature Field**

Defines a temperature field for the three-node beam element (CBEAM3 entry).

**Format:**

| 1      | 2      | 3       | 4      | 5     | 6     | 7      | 8      | 9      | 10 |
|--------|--------|---------|--------|-------|-------|--------|--------|--------|----|
| TEMPB3 | SID    | EID     | T(A)   | T(B)  | T(C)  | TPY(A) | TPZ(A) | TPY(B) |    |
|        | TPZ(B) | TPY(C)  | TPZ(C) | TC(A) | TD(A) | TE(A)  | TF(A)  | TC(B)  |    |
|        | TD(B)  | TE(B)   | TF(B)  | TC(C) | TD(C) | TE(C)  | TF(C)  |        |    |
|        |        | Element | ID     | List  |       |        |        |        |    |

**Example:**

|        |      |     |      |      |      |      |      |      |  |
|--------|------|-----|------|------|------|------|------|------|--|
| TEMPB3 | 101  | 23  | 45.9 | 10.0 | 0.0  | 1.3  | 23.9 | 3.8  |  |
|        |      | 2.5 | 68.0 | 91.0 | 45.0 |      | 48.0 | 80.0 |  |
|        | 20.0 |     | 33.9 |      |      | 45.6 |      |      |  |
|        | 9    | 10  | THRU | 30   | 41   | 51   | 67   | 78   |  |
|        | THRU | 110 | BY   | 2    |      |      |      |      |  |

| Descriptor      | Meaning   |
|-----------------|---|
| SID             | Temperature set identification number. (Integer > 0; Required)  |
| EID             | Element identification number. (Integer > 0; Required)  |
| T( <i>j</i> )   | Temperature at <i>j</i> ( <i>j</i> =A,B,C) on the neutral axis. (Real; Default = 0.0)   |
| TP <i>i(j</i> ) | Effective linear gradient in local direction <i>i</i> ( <i>i</i> = y, z) at <i>j</i> ( <i>j</i> = A, B, C). (Real; Default = 0.0)                                       |
| T <i>i(j</i> )  | Temperature at stress recovery point <i>i</i> ( <i>i</i> =C, D, E, F) defined in PBEAM3 at location <i>j</i> ( <i>j</i> =A, B, C). (Real; Default = 0.0; see Remark 3.) |
| Element ID      | List of CBEAM3 element identification numbers. Character strings "THRU" and "BY"  |
| List            | may be used in the list. (Integer > 0; "THRU" or "BY". At least one element ID is required.)  |

**Remarks:**

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOAD*i* or TLOAD*i* entry.
2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If all T*i(j*) fields are blank, linear temperature gradients are assumed for stress recovery.

4. Temperature field defined by TEMPB3 entry always takes precedence over the grid point temperatures given by TEMP and TEMPD entries.
5. The effective thermal gradients are defined in the local coordinate system. For their definitions, see Remark 6 of Bulk Data entry TEMPRB for the details.

**TEMPBC****Grid Point Temperatures**

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions (SOLs 153 and 159 only).

**Format:**

| 1      | 2   | 3    | 4     | 5    | 6     | 7    | 8     | 9    | 10 |
|--------|-----|------|-------|------|-------|------|-------|------|----|
| TEMPBC | SID | TYPE | TEMP1 | GID1 | TEMP2 | GID2 | TEMP3 | GID3 |    |

**Example:**

|        |    |      |       |   |       |   |       |   |  |
|--------|----|------|-------|---|-------|---|-------|---|--|
| TEMPBC | 10 | STAT | 100.0 | 1 | 100.0 | 2 | 100.0 | 3 |  |
|--------|----|------|-------|---|-------|---|-------|---|--|

**Alternate Format and Example:**

|        |     |      |       |      |        |      |      |     |  |
|--------|-----|------|-------|------|--------|------|------|-----|--|
| TEMPBC | SID | TYPE | TEMP1 | GID1 | "THRU" | GID2 | "BY" | INC |  |
|--------|-----|------|-------|------|--------|------|------|-----|--|

|        |    |      |       |   |      |    |    |   |  |
|--------|----|------|-------|---|------|----|----|---|--|
| TEMPBC | 20 | STAT | 100.0 | 4 | THRU | 50 | BY | 2 |  |
|--------|----|------|-------|---|------|----|----|---|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Temperature set identification number. (Integer > 0)  |
| TYPE       | Type of temperature boundary condition. See Remarks. (Character; Default = "STAT"):<br>STAT - Constant temperature boundary condition<br>TRAN - Time-varying temperature boundary condition |
| TEMPi      | Temperature (Real)  |
| GIDI       | Grid point identification number. (Integer>0 or "THRU" or "BY")   |
| INC        | Grid point number increment. (Integer)  |

**Remarks:**

- For a constant Boundary Condition (TYPE = "STAT"), the temperature boundary load set (SID) is selected in the Case Control Section (SPC = SID). TYPE = "STAT" may be used in both steady-state (SOL 153) and transient analysis (SOL 159).
- For transient analysis (SOL 159), a constant boundary condition should be specified using the SPC Bulk Data entry.
- For a time-varying boundary condition (TYPE = "TRAN"), SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. TYPE="TRAN" is permitted only in transient analysis (SOL 159). A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  provides any required time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis.

4. In the alternate format, TEMP1 is the nodal temperature for the grid points GID1,GID1+INC,...,GID2. If "BY" and INC are not specified, then the grid point number increment is unity.
5. If TYPE = "STAT", then no SPCi Bulk Data entries may be specified.
6. If TYPE = "TRAN", then no CELAS2 or DAREA Bulk Data entries may be specified. Also, "U" must be specified in the CONV field on the entry to obtain accurate results.
7. All TEMPBC entries in the Bulk Data Section must indicate either TYPE = "STAT" or TYPE = "TRAN" but not both.
8. In transient thermal analysis, the TEMPBC option is used to set a grid, known temperature as a function of time. Internally NASTRAN uses SLOAD and CELAS2 entries to enforce the temperature as a function of time. The  $u=P/K$  or temperature is equal to SLOAD divided by CELAS2. The default stiffness for the CELAS2 entry is 1.0E10. This value is fine most of the time. However, if the user desired to run the model using thermal conductivity in the following unit (Btu/sec.inch.F), then it may run into a numerically convergence issue. This is because the thermal conductivity for this unit has conductivity value in the 1.0E-6 range. The avoidance is to set a NASTRAN system cell, TBCMAG to 1.0E2.
9. The TEMPBC with type=TRAN is no longer supported in SOL 400 transient thermal analysis. One should use the SPC1 and SPCD to enforced temperature. For example, convection coefficient as a function of time, mass flow rate as a function of time, or ambient temperature as a function of time all used the TEMPBC,TRAN option to apply a time varying nodal quantity in SOL 159. Following are the procedures for a user to convert SOL 159 into SOL 400 when there is TEMPBC,TRAN entries.

The SPC entry used to fixed a boundary temperature at a particular value at all time. However, if you have a TEMPBC,TRAN in your test file this means that all the permanent SPC must be converted into SPCD and SPC1 with a unit step function on the TABLED1 in SOL 400. The avoidance to this, using a constant temperature with SPC and a time-varying temperature in the same run, is using the large stiffness method to enforced the time varying quantity. The large stiffness used SLOAD and CELAS2 which  $u = P / K$  which  $P = \text{SLOAD}$ , and  $K = \text{CELAS2}$ , and  $u$  is the desired temperature times the time-varying quantity in the TABLEDx.

To Convert SOL 159 Models to SOL 400 Models:

- a. Executive Control Section - change SOL 159 to SOL 400.
- b. Case Control Section - replace ANALYSIS=HEAT by ANALYSIS=HTRAN, also add SPC if all temperature boundary conditions are transient (the following Case 3b).
- c. Bulk Data Section - replace the "TRAN" type TEMPBC by SPC1 and SPCD. The details are explained below.

If all temperature boundary conditions are constant, no changes are required.

If all temperature boundary conditions are transient, replace TEMPBC by SPC1 and SPCD and modify TLOAD1.

For example, replace the following entries of SOL 159 model:

TLOAD1,40,400,,4000  
TEMPBC,400,TRAN,300.0,99

by

SPC = 111 (Case CC)  
:  
**TLOAD1,40,400,,1,4000**  
SPCD,400,99,,300.0  
SPC1,111,,99

If a model has both constant and transient temperature boundary conditions, all boundary conditions must be converted into SPC1 and SPCD.

For example, replace the following entries of SOL 159 model:

DLOAD,222,1.0,1,0,30,1.0,40  
**TLOAD1,40,400,,4000**  
TEMPBC,400,TRAN,300.0,99  
SPC,111,98,,20.0

by

DLOAD,222,1.0,1,0,30,1.0,40, **1.0,50**  
**TLOAD1,40,400,,1,4000**  
SPCD,400,99,,300.0  
SPC1,111,,99  
**TLOAD1,50,500,,1,5000**  
SPCD,500,98,,20.0  
SPC1,111,,98  
TABLED1,5000,,,,  
,0.0,1.0,1000.0,1.0,ENDT

10. If TYPE=TRAN and the initial temperature is non zero, then the initial temperature must be supplied. Use case control IC and bulk data TEMP entries.

**TEMPD**

## Grid Point Temperature Field Default

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP or TEMPN1 (for heat transfer analysis) entries.

Format:

| 1     | 2    | 3  | 4    | 5  | 6    | 7  | 8    | 9  | 10 |
|-------|------|----|------|----|------|----|------|----|----|
| TEMPD | SID1 | T1 | SID2 | T2 | SID3 | T3 | SID4 | T4 |    |

Example:

|       |   |       |  |  |  |  |  |  |  |
|-------|---|-------|--|--|--|--|--|--|--|
| TEMPD | 1 | 216.3 |  |  |  |  |  |  |  |
|-------|---|-------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SIDi       | Temperature set identification number. (Integer > 0) |
| Ti         | Default temperature value. (Real)                    |

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to four default temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data is defined.
7. For steady-state heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperatures boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC=SID request selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
10. For partitioned Bulk Data superelements and auxiliary models, TEMPD must be specified in all partitioned Bulk Data Sections.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TEMPF****p-Element Temperature Field with Function Definition**

Defines the thermal loading to be applied to a group of elements.

**Format:**

| 1     | 2    | 3    | 4      | 5      | 6 | 7 | 8 | 9 | 10 |
|-------|------|------|--------|--------|---|---|---|---|----|
| TEMPF | SID  | EID1 | FTEMP  | FTABID |   |   |   |   |    |
|       | EID2 | EID3 | -etc.- |        |   |   |   |   |    |

**Example:**

|       |     |    |     |  |  |  |  |  |  |
|-------|-----|----|-----|--|--|--|--|--|--|
| TEMPF | 127 | 12 | 111 |  |  |  |  |  |  |
|-------|-----|----|-----|--|--|--|--|--|--|

**Alternate Format:**

|       |      |        |       |        |  |  |  |  |  |
|-------|------|--------|-------|--------|--|--|--|--|--|
| TEMPF | SID  | EID1   | FTEMP | FTABID |  |  |  |  |  |
|       | EID2 | “THRU” | EIDn  |        |  |  |  |  |  |

| Descriptor | Meaning   | Type        | Default  |
|------------|---|-------------|----------|
| SID        | Temperature set identification number.  | Integer > 0 | Required |
| FTEMP      | ID of a DEQATN entry describing the temperature field as a function of x,y,z. See Remark 1. | Integer > 0 |          |
| FTABID     | ID of a TABLE3D entry describing the temperature field. See Remark 1.                       | Integer > 0 |          |
| EIDi       | Identification numbers of the p-elements to which this thermal load is applied.             | Integer > 0 | Required |

**Remarks:**

1. Either FTEMP or FTABID must be specified but not both.
2. The TEMPF entry overrides the temperature at the element vertices specified on the TEMP or TEMPD entries.

**TEMPN1**

## TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400

Defines initial temperature at grid points of heat shell elements with linear or quadratic temperature distribution across the thickness direction.

Format:

| 1      | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|-----|----|----|----|----|----|----|---|----|
| TEMPN1 | SID | G1 | C1 | T1 | G2 | C2 | T2 |   |    |

Example:

|        |    |     |     |       |  |  |  |  |  |
|--------|----|-----|-----|-------|--|--|--|--|--|
| TEMPN1 | 10 | 100 | 123 | 1300. |  |  |  |  |  |
|--------|----|-----|-----|-------|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Temperature set identification number. (Integer > 0)  |
| Gi         | Grid point identification number. (Integer > 0)   |
| Ci         | Component numbers. ( $0 \leq \text{Integer} \leq 3$ ; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1) |
| Ti         | Temperature. (Real)   |

Remarks:

1. This entry is for shell elements defined on a PSHLN1 heat transfer.
2. In the steady-state solution sequences, SID is selected by the LOAD Case Control command.
3. In the transient solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
4. The TEMPN1 Bulk Data entry is used to complement the TEMP Bulk Data entry. The TEMP entry will just initiate the TOP grids. This entry allows specifying of the TOP, BOT, and MID or any combination. See the NLMOPTS, TEMGO, vmaptg Bulk Data entry on how to list internally generated grids.
5. For steady-state heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT)=SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
6. For transient heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initial condition temperatures. The Case Control command IC=SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.

**TEMPP1****Plate Element Temperature Field, Form 1**

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading, temperature-dependent material properties, or stress recovery.

**Format:**

| 1      | 2    | 3    | 4    | 5      | 6    | 7    | 8      | 9 | 10 |
|--------|------|------|------|--------|------|------|--------|---|----|
| TEMPP1 | SID  | EID1 | TBAR | TPRIME | T1   | T2   |        |   |    |
|        | EID2 | EID3 | EID4 | EID5   | EID6 | EID7 | -etc.- |   |    |

**Example:**

|        |    |    |      |      |      |      |  |  |  |
|--------|----|----|------|------|------|------|--|--|--|
| TEMPP1 | 2  | 24 | 62.0 | 10.0 | 57.0 | 67.0 |  |  |  |
|        | 26 | 21 | 19   | 30   |      |      |  |  |  |

**Alternate Format and Example of Continuation Entry:**

|  |      |        |      |      |        |      |  |  |  |
|--|------|--------|------|------|--------|------|--|--|--|
|  | EID2 | “THRU” | EIDi | EIDj | “THRU” | EIDk |  |  |  |
|  | 1    | THRU   | 10   | 30   | THRU   | 61   |  |  |  |

| Descriptor          | Meaning   |
|---------------------|---|
| SID                 | Temperature set identification number. (Integer > 0)  |
| EIDi, EIDj,<br>EIDk | Unique element identification number(s). (Integer > 0 or the continuation entries may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi and EIDj < EIDk.)   |
| TBAR                | Temperature at the element’s reference plane as defined by ZOFFS on the connection entry. (Real, Default 0.0)   |
| TPRIME              | Effective linear thermal gradient. Not used for membranes. (Real, Default 0.0)  |
| T1, T2              | Temperatures for stress calculation at points defined on the element property entry. ( $Z_1$ and $Z_2$ field on PSHELL entry.) T1 may be specified on the lower surface and T2 on the upper surface for the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR elements. These data are not used for membrane elements. See Remark 9. If both T1 and T2 are blank, they are computed from the equation $T = \text{TBAR} + z \cdot \text{TPRIME}$ , where $z$ is the distance from the center fiber. The program replaces T1 with a flag, and $z$ is computed in a later operation. (Real) |

**Remarks:**

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry. TBAR and TPRIME are used for the analysis. If both are left blank the elements in essence see no thermal loading. T1 and T2 are used for post analysis for stress calculations.

2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If continuation entries are present, EID1 and elements specified on the continuation entry are used. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. For temperature field other than a constant gradient, the “effective gradient” for a homogeneous plate is

$$\text{TPRIME} = \frac{1}{I} \int_z T(z) z dz$$

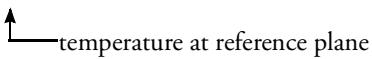
where  $I$  is the bending inertia and  $z$  is the distance from the neutral surface in the positive normal direction.

6. The “average” temperature for a homogeneous plate is

$$\text{TBAR} = \frac{1}{\text{Volume}} \int_{\text{Volume}} T d\text{Volume}$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature TBAR.
8. Large “THRU” ranges will lead to System Fatal Message 3008 (“Insufficient Core”) and should be avoided, particularly for open sets.
9. If the element material is nonlinear then T1 and T2 should be left blank (see the MATS1 entry).
10. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
11. The bending and twisting moments can be reduced to outer fiber stresses and combined with membrane stresses in the composite plate elements. If, in addition, the temperature is specified by the user at a point where outer fiber stresses are calculated, the thermal expansion due to the difference between the specified temperature and the temperature that would be produced by a uniform gradient,  $T'$ , is assumed to be completely restrained. Stated differently, the second and higher order moments of the thermal expansion are assumed to be completely restrained by elastic stiffness. The resulting stress increment is

$$\{\Delta\sigma\} = -[G_e]\{\alpha_e\}(T - T_o - T'z)$$



temperature at reference plane

where  $[G_e]$  and  $\{\alpha_e\}$  are evaluated for the average temperature of the element  $\bar{T}$ .

[TEMPP3](#)

Plate Element Temperature Field, Form 3

TEMPP3 is no longer available. Use TEMPP1.

**TEMPPRB****One-Dimensional Element Temperature Field**

Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

**Format:**

| 1       | 2    | 3    | 4    | 5    | 6    | 7    | 8      | 9    | 10 |
|---------|------|------|------|------|------|------|--------|------|----|
| TEMPPRB | SID  | EID1 | TA   | TB   | TP1A | TP1B | TP2A   | TP2B |    |
|         | TCA  | TDA  | TEA  | TFA  | TCB  | TDB  | TEB    | TFB  |    |
|         | EID2 | EID3 | EID4 | EID5 | EID6 | EID7 | -etc.- |      |    |

**Example:**

|         |      |      |      |      |      |      |      |     |  |
|---------|------|------|------|------|------|------|------|-----|--|
| TEMPPRB | 200  | 1    | 68.0 | 23.0 | 0.0  | 28.0 |      | 2.5 |  |
|         | 68.0 | 91.0 | 45.0 |      | 48.0 | 80.0 | 20.0 |     |  |
|         | 9    | 10   |      |      |      |      |      |     |  |

**Alternate Format and Example of Continuation Entry:**

|  |      |        |      |      |        |      |  |  |  |
|--|------|--------|------|------|--------|------|--|--|--|
|  | EID2 | "THRU" | EIDi | EIDj | "THRU" | EIDk |  |  |  |
|  | 2    | THRU   | 4    | 10   | THRU   | 14   |  |  |  |

| Descriptor          | Meaning   |
|---------------------|---|
| SID                 | Temperature set identification number. (Integer > 0)  |
| EIDi, EIDj,<br>EIDk | Unique element identification number(s). (Integer > 0 or the second continuation entry may have "THRU" in fields 3 and/or 6 in which case EID2 < EIDi and EIDj < EIDk.)   |
| TA, TB              | Temperature at end A and end B on the neutral axis. (Real, default 0.0)   |
| TPij                | Effective linear gradient in direction i on end j; used with CBAR, CBEAM, and CBEND only. (Real)  |
| Tij                 | Temperature at point i (i=C, D, E, or F) as defined on the PBAR, PBEAM, and PBEND entries at end j (j=A or B). This data is used for stress recovery only with CBAR, CBEAM, and CBEND exclusively. See Remark 3. (Real) |

**Remarks:**

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. If at least one nonzero or nonblank  $T_{ij}$  is present, the point temperatures given are used for stress recovery. If no  $T_{ij}$  values are given, linear temperature gradients are assumed for stress recovery. The  $T_{ij}$  values are not used in the calculation of differential stiffness. The default for  $T_{ij}$  for  $j=A$  is  $TA$  and the default for  $T_{ij}$  for  $j=B$  is  $TB$ .
4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used. Elements must not be specified more than once.
5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. The effective thermal gradients in the element coordinate system for the CBAR element are defined by the following integrals over the cross section. For end "A" (end "B" is similar),

$$TA = \frac{1}{A} \int TA(y, z) dA$$

$$TP1A = \frac{I_2}{\Delta} \int_A (y - y_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (z - z_n) TA(y, z) dA$$

$$TP2A = \frac{I_1}{\Delta} \int_A (z - z_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (y - y_n) TA(y, z) dA$$

$$\Delta = I_1 I_2 - I_{12}^2$$

if  $I_{12} = 0$

$$TP1A = \frac{1}{I_1} \int_A (y - y_n) TA(y, z) dA$$

$$TP2A = \frac{1}{I_2} \int_A (z - z_n) TA(y, z) dA$$

where  $TA(y, z)$  is the temperature at point  $y, z$  (in the element coordinate system) at end "A" of the bar. See the CBAR entry description for the element coordinate system:  $I_1$ ,  $I_2$  and  $I_{12}$  are the moments of inertia about the  $z$  and  $y$  axes, respectively. The temperatures are assumed to vary linearly along the length (x-axis). Note that if the temperature varies linearly over the cross section, then TP1A, TP1B, TP2A and TP2B are the actual gradients.

7. If the element material is temperature-dependent, the material properties are evaluated at the average temperature  $(TA + TB)/2$ .
8. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS, 417](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
9. If any  $T_y$  is specified the stresses computed by the effective gradient are corrected by  $\Delta\sigma$  such that:

$$\sigma = \sigma|_{T_A + y^{TP1A} + z^{TP2A}} + \Delta\sigma$$

where  $\Delta\sigma$  is in the form

$\Delta\sigma = -\alpha E [T_{CA} - T_o - C_1 \cdot TP1A - C_2 \cdot TP2A]$  etc

for CBAR and CBEAM

$\Delta\sigma = -\alpha E [T_{CA} - T_o - (C_1 + \Delta N) \cdot TP1A - C_2 \cdot TP2A]$  etc

for CBEND.

**TERMIN**

Control to Terminate a SOL 600 Analysis Under Certain Conditions

Used in SOL 600 only.

Format:

| 1      | 2     | 3   | 4     | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|-------|-----|-------|-----|---|---|---|---|----|
| TERMIN | ID    | NC  |       |     |   |   |   |   |    |
|        | NTYPE | NBN | ICRIT | VAL |   |   |   |   |    |

Example:

|        |   |      |    |     |  |  |  |  |  |
|--------|---|------|----|-----|--|--|--|--|--|
| TERMIN | 2 | 2    |    |     |  |  |  |  |  |
|        | 7 | 1000 | -1 | 2.0 |  |  |  |  |  |
|        | 7 | 1000 | -2 | 0.8 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | ID corresponding to a Case TERMIN entry. (Integer > 0; no Default)   |
| NC         | Number of termination conditions to be specified. (Integer > 0; Default = 1; Max number is 10)   |
| NTYPE      | <p>Termination Criteria Type (Integer; no Default)</p> <p>Enter 1 if termination occurs when a percentage of the boundary nodes are in contact.</p> <p>Enter 2 if termination occurs when the maximum force on a rigid body is exceeded.</p> <p>Enter 3 if termination occurs when the displacement of the rigid body exceeds the allowed displacement.</p> <p>Enter 5 if termination occurs when the distance between the reference points of two rigid bodies is less or greater than the specified value.</p> <p>Enter 6 if termination occurs, when any displacement in body, is greater than the specified value.</p> <p>Enter 7 if termination occurs, when the displacement in the node, is greater than the specified value.</p> |
| NBN        | Body number, for criterion type 7, grid ID (Integer > 0; no Default)   |

| Descriptor | Meaning  |
|------------|--|
| ICRIT      | <p>Criteria specification. (Integer; no Default)</p> <p>For criterion type 1, enter the percentage of nodes to be in contact for termination; default = 100.</p> <p>For criterion type 2, enter direction 1/2/3 for the x, y, z global directions</p> <p>For criterion type 5, enter the second body.</p> <p>For criterion type 6 or 7, enter the degree of freedom.</p> <p>For criterion type 6 or 7, enter -1 if the total translational displacement.</p> <p>For criterion type 6 or 7, enter -2 if the total rotation.</p> |
| VAL        | <p>Termination value. (Real; no Default)</p> <p>For criterion type 2, enter the critical force.</p> <p>For criterion type 3, enter the critical maximum displacement.</p> <p>For criterion type 5, enter the critical distance. If the value is positive, the termination occurs when the distance is less than the value. If the value is negative, the termination occurs when the distance is greater than the value in a positive sign.</p> <p>For criterion type 6 or 7, enter the critical distance (rotation).</p>      |

#### Remarks:

1. Different TERMIN entries may be used in different subcases.
2. Not all subcases require TERMIN entries if used in other subcases.

## TF

## Dynamic Transfer Function

Defines a dynamic transfer function of the form

$$(B_0 + B_1 \cdot p + B_2 \cdot p^2)u_d + \sum_i (A_0(i) + A_1(i)p + A_2(i)p^2)u_i = 0 \quad (9-34)$$

Where:

$U_d$  = dependent coordinate

$U_i$  = independent degree of freedom

$p$  = differential operator ( $p=d/dt$ )

Can also be used as a means of direct matrix input. See Remark 4.

Format:

| 1  | 2    | 3    | 4     | 5     | 6     | 7      | 8 | 9 | 10 |
|----|------|------|-------|-------|-------|--------|---|---|----|
| TF | SID  | GD   | CD    | B0    | B1    | B2     |   |   |    |
|    | G(1) | C(1) | A0(1) | A1(1) | A2(1) | -etc.- |   |   |    |

Example:

|    |   |   |     |     |     |     |  |  |  |
|----|---|---|-----|-----|-----|-----|--|--|--|
| TF | 1 | 2 | 3   | 4.0 | 5.0 | 6.0 |  |  |  |
|    | 3 | 4 | 5.0 | 6.0 | 7.0 |     |  |  |  |

| Descriptor          | Meaning  |
|---------------------|--|
| SID                 | Set identification number. (Integer > 0)   |
| GD, G(i)            | Grid, scalar, or extra point identification numbers. (Integer > 0)   |
| CD, C(i)            | Component numbers. (Integer zero or blank for scalar or extra points, any one of the Integers 1 through 6 for a grid point.) |
| B0, B1, B2          | Transfer function coefficients. (Real)   |
| A0(i), A1(i), A2(i) |  |

Remarks:

1. Transfer function sets must be selected with the Case Control command TFL = SID.
2. Continuation entries are optional.
3. The matrix elements defined by this entry are added to the dynamic matrices for the problem.
4. The constraint relation given in Eq. (9-34) will hold only if no structural elements or other matrix elements are connected to the dependent coordinate  $u_d$ . In fact, the terms on the left side of Eq. (9-34) are simply added to the terms from all other sources in the row for  $u_d$ .

5. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of transfer functions.
6. For each SID, only one logical entry is allowed for each GD, CD combination.
7. For heat transfer analysis, the initial conditions must satisfy Eq. [\(9-34\)](#).
8. RC network does not support TF for thermal analysis.
9. For more information see [DMIGs, Extra Points, and Transfer Functions](#) (p. 658) in the *Dynamic Analysis User's Guide*.

**THPAD****ROMAC's THPAD User Defined Service Element Property**

Allows the user to provide the parameters for a tilting pad bearing for use with ROMAC's THPAD service.

**Format:**

| 1      | 2           | 3       | 4        | 5            | 6      | 7      | 8      | 9     | 10 |
|--------|-------------|---------|----------|--------------|--------|--------|--------|-------|----|
| THPAD  | RID         | TLIMIT  |          |              |        |        |        |       |    |
| Title1 |             |         |          |              |        |        |        |       |    |
| Title2 |             |         |          |              |        |        |        |       |    |
|        | RADIUS      | CLEAR   | OUTR     | LENGT<br>H   | ALPHP  | E      | ALPHJ  | ALPHS |    |
|        | NPADS       | NEL     | IECC     |              |        |        |        |       |    |
|        | ARC1        | OFFSET1 | PRELOAD1 | PVANG1       | IP1    | KP1    | MP1    | IT1   |    |
|        | ARC2        | OFFSET2 | PRELOAD  | PVANG2       | IP2    | KP2    | MP2    | IT2   |    |
|        | ...         | ...     | ...      | ...          | ...    | ...    | ...    | ...   |    |
|        | KFILM       | KPAD    | TBACK    | TJF          | PSUMP  | TIN    | KCAV   | CCAV  |    |
|        | DENSIT<br>Y | SPEC    | TA       | MUA          | TB     | MUB    | ESUMP  | TMANU |    |
|        | ERROR       | XG      | YG       |              |        | NAX    | FACTOR | XFACT |    |
|        | ITJ         | IBC     | ITUR     | IDIM         |        | ICOND  | ITB    | ITG   |    |
|        |             |         | ICAV     | ICROSS       | IDEF   | IFLEX  | IUN    | ISUMP |    |
|        | KTHETA      | DEL0    | HOTOVER  | COLDOVE<br>R |        |        |        |       |    |
|        | NCASE       | MAXC    | IAPH     |              |        |        |        |       |    |
|        | RPM1        | FX1     | FY1      | PGNU1        | RELAX1 | OFLOW1 | DEREL1 |       |    |
|        | RPM2        | FX2     | FY2      | PGNU2        | RELAX2 | OFLOW2 | DEREL2 |       |    |
|        | ...         | ...     | ...      | ...          | ...    | ...    | ...    | ...   |    |

**Example:**

In the FMS Section of the Nastran input stream a connect statement is required:

```
CONNECT SERVICE GR1 'SCR.MDSolver.Obj.Uds.Elements.thpad'
```

In Bulk Data:

|         |   |         |     |       |      |    |  |  |  |
|---------|---|---------|-----|-------|------|----|--|--|--|
| CBUSH2D | 100   | 200     | 1   | 2     |      | XY |  |  |  |
| PBUSH2D | 200   |         |     |       |      |    |  |  |  |
| ELEMUDS | 200   | PBUSH2D | GR1 | THPAD | FREQ |    |  |  |  |
| THPAD   | 200   | 1000.0  |     |       |      |    |  |  |  |
|         | Tilting pad bearing used for the 8 stage centrifugal compressor |         |     |       |      |    |  |  |  |

|  | Load on Pad |         |       |         |       |        |        |        |  |
|--|-------------|---------|-------|---------|-------|--------|--------|--------|--|
|  | 2.5         | 0.0038  | 3.5   | 3.0     | 3.0E7 | 6.5E-6 | 6.5E-6 | 6.5E-6 |  |
|  | 5           | 30      | 0     |         |       |        |        |        |  |
|  | 60.0        | 0.5     | 0.3   | 54.0    | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 126.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 198.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60          | 0.5     | 0.3   | 270.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 342.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 2.0E-6      | 0.00067 | 120.0 | 175.0   | 0.0   | 175.0  | 5.0    | 0.0    |  |
|  | 8.0E-5      | 180.0   | 100.0 | 4.13E-6 | 210.0 | 6.8E-7 | 1.0    | 75.0   |  |
|  | 0.001       | 0.2     | 0.2   |         |       | 2.0    | 100.0  | 100.0  |  |
|  | 0           | 2       | 1     | 0       |       | 0      | 0      | 0      |  |
|  |             |         | 1     | 0       | 2     | 0      | 0      | 0      |  |
|  | 0.0         | 0.0     | 100.0 | 0.0     |       |        |        |        |  |
|  | 2           | 100     | 100   |         |       |        |        |        |  |
|  | 1000.0      | 0.0     | 931.5 | 1.0     | 1.0   | 0.0    | 1.0    |        |  |
|  | 2000.0      | 0.0     | 931.5 | 1.0     | 1.0   | 0.0    | 1.0    |        |  |

| Descriptor | Meaning   |
|------------|---|
| RID        | ROMAC bearing identification number that matches the property identification number on an ELEMUDS entry as well as a PBUSH2D entry. (Integer > 0) |
| TLIMIT     | Time limit for a call to THPAD. If the call takes longer than this, the service call will be killed. (Real)                                       |
| TITLE1     | User supplied title. (Character)  |
| TITLE2     | User supplied title. (Character)  |
| RADIUS     | Journal radius (Real, in)   |
| CLEAR      | Ground-in pad clearance. (Real, in)   |
| OUTR       | Outer radius of bearing pads. (Real, in)  |
| LENGTH     | Axial length of bearing pads. (Real, in)  |
| E          | Young's modulus for the pads. (Real, psi)   |
| ALPHP      | Pad thermal expansion coefficient. (Real, 1/F)  |
| ALPHJ      | Journal thermal expansion coefficient. (Real, 1/F)  |
| ALPHS      | Shell thermal expansion coefficient. (Real, 1/F)  |
| NPADS      | Number of pads. (Integer > 0)   |
| NEL        | Number of elements per pad. (Integer > 0)   |
| IECC       | Eccentricity variation flag. (Integer)  |
| 0          | Allow eccentricity variation to match load.   |

| Descriptor | Meaning   |
|------------|---|
| 1          | Eccentricity fixed at initial value.                              |
| ARCi       | Arc length for the pad. (Real, deg)                               |
| OFFSETi    | Offset factor for the pad. (Real)                                 |
| PRELOADi   | Preload factor for the pad. (Real)                                |
| PVANGi     | Location of pad pivot angle. (Real, deg)                          |
| IPi        | Pad polar moment. (Real, lb-sec <sup>2</sup> -in)                 |
| KPi        | Pad pivot stiffness. KPi=0 for rigid pivots. (Real, lb/in)        |
| MPi        | Pad mass. (Real, lb-sec <sup>2</sup> /in)                         |
| ITi        | Pad bending inertia. (Real, in <sup>4</sup> )                     |
| KFILM      | Fluid thermal conductivity. (Real, btu/in-sec-F)                  |
| KPAD       | Pad thermal conductivity. (Real, btu/in-sec-F)                    |
| TBACK      | Back of pad temperature. (Real, F)                                |
| TJF        | Temperature of the journal surface. (Real, F)                     |
| PSUMP      | Sump oil pressure. (Real, psi)                                    |
| TIN        | Pad inlet oil temperature. (Real, F)                              |
| KCAV       | Thermal conductivities cavitation ratio. (Real)                   |
| CCAV       | Cavitation latent heat ratio. (Real, btu/sec-in <sup>2</sup> -F)  |
| DENSITY    | Lubricant density. (Real, lbf-sec <sup>2</sup> /in <sup>4</sup> ) |
| SPEC       | Lubricant specific heat. (Real, btu-in/lbf-sec <sup>2</sup> -F)   |
| TA         | First temperature datum. (Real, F)                                |
| MUA        | Viscosity at TA. (Real)   |
| TB         | Second temperature datum. (Real, F)                               |
| MUB        | Viscosity at TB. (Real)   |
| ESUMP      | Sump heating factor. (Real)                                       |
| TMANU      | Clearance set temperature. (Real, F)                              |
| ERROR      | Error criterion. (Real)   |
| XG         | Initial guess of x/cp. (Real)                                     |
| YG         | Initial guess of y/cp. (Real)                                     |
| NAX        | Axial pressure exponent. (Real, 2.0 is recommended)               |
| FACTOR     | Perturbation effect on position iteration. (Real)                 |
| XFACT      | Percentage of cross-coupled terms in position update. (Real)      |
| ITJ        | Journal temperature flag. (Integer)                               |
| 0          | Journal temperature found as average of film temperature.         |

| Descriptor | Meaning  |
|------------|--|
|            | 1 Journal temperature fixed by user.<br>2 Journal temperature set for zero heat flux to shaft.   |
| IBC        | Boundary condition flag. (Integer)<br>0 Reynolds boundary condition with flow correction.<br>1 Not used.<br>2 Reynolds boundary condition for pressure.                  |
| ITUR       | Turbulence flag. (Integer)<br>0 Laminar solution.<br>1 Turbulence allowed.   |
| IDIM       | Dimension flag. (Integer)<br>0 Two-dimensional conduction in the pad.<br>1 Radial conduction in the pad.   |
| ICOND      | Conduction flag. (Integer)<br>0 Energy equation includes conduction.<br>1 Adiabatic (isothermal) energy equation.  |
| ITB        | Back temperature flag. (Integer)<br>0 Pad back temperature equals sump temperature.<br>1 Pad back temperature fixed by user.   |
| ITG        | Groove temperature flag. (Integer)<br>0 Groove temperature found iteratively by heat balance.<br>1 Groove temperature fixed at sump temperature.                         |
| ICAV       | Cavitation flag. (Integer)<br>0 No cavitation effects in energy equation.<br>1 Include cavitation effects in energy equation.  |
| ICROSS     | Cross-film viscosity flag. (Integer)<br>0 Constant cross-film viscosity.<br>1 Variable cross-film viscosity.   |
| IDEF       | Deformation flag. (Integer)<br>0 No pad/pivot deformations.<br>1 Pivot deformations.<br>2 Pad, journal and shell thermal deformations.<br>4 Pad mechanical deformations. |
| IFLEX      | Type of beam bending for pad deformations. (Integer)   |

| Descriptor | Meaning   |
|------------|---|
|            | 0 Curved beam analysis.<br>1 Straight beam analysis.<br>2 Simple curvature calculation.   |
| IUN        | Unloaded pads flag. (Integer)<br>0 Effect of unloaded pads is considered.<br>1 Effect of unloaded pads is neglected.                              |
| ISUMP      | Sump temperature flag. (Integer)<br>0 Sump temperature set to pad inlet oil temperature.<br>1 Sump temperature found iteratively by heat balance. |
| KTHETA     | Pivot rotational stiffness. (Real, lb-in, Default = 0.0)  |
| DEL0       | Initial pad angle for moment balance. (Real, rad, Default = 0.0)  |
| HOTOVER    | Percentage of hot oil available carried to the groove. (Real, Default = 100.0)  |
| COLDOVER   | Percentage of groove oil made up of inlet oil. (Real, Default = 0.0)  |
| NCASE      | Number of speed/load cases. (Integer > 0)   |
| MAXC       | Maximum number of position iterations. (Real, Default = 100)  |
| IALPH      | New stiffness smoothing factor. (Real)  |
| RPMi       | Speed of journal. (Real, rpm)   |
| FXi        | Applied force in the negative x-direction. (Real, lb)   |
| FYi        | Applied force in the negative y-direction. (Real, lb)   |
| PGNUi      | Whirl ratio of shaft. (Real)  |
| RELAXi     | Position iteration relaxation factor. (Real)  |
| OFLOWi     | Oil flow to bearing. (Real, cips)   |
| DERELi     | Relaxation factor for pivot deformation iteration. (Real)   |

### Remarks:

1. This entry triggers the call to the THPAD service. The GROUP must match the GROUP field of the CONNECT SERVICE FMS entry.
2. On the FMS CONNECT entry, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing ELEMUDS and PBUSH2D PIDs.
4. All units must be in the English system.
5. Refer to <http://www.virginia.edu/romac/> for more information.

6. Use of this entry requires, that the user, has obtained the source for the THPAD routine from the *University of Virginia Rotating Machinery and Controls Laboratory (ROMAC)* and use the MSC Nastran Software Development Kit (SDK) to build it as a User Defined Service (UDS). SDK and UDS build instructions are available in the [MSC Nastran Rotordynamics User's Guide](#).

**TIC****Transient Analysis Initial Condition**

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom (See Remark 6.). This entry may not be used for heat transfer analysis.

**Format:**

| 1   | 2   | 3 | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|-----|-----|---|---|----|----|---|---|---|----|
| TIC | SID | G | C | U0 | V0 |   |   |   |    |

**Example:**

|     |     |    |   |     |     |  |  |  |  |
|-----|-----|----|---|-----|-----|--|--|--|--|
| TIC | 100 | 10 | 3 | 0.1 | 0.5 |  |  |  |  |
|-----|-----|----|---|-----|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| G          | Grid, scalar, or extra point or modal coordinate identification number. (Integer > 0). See Remark 4.   |
| C          | Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.) |
| U0         | Initial displacement. (Real)   |
| V0         | Initial velocity. (Real)   |

**Remarks:**

1. Transient analysis initial condition sets must be selected with the IC Case Control command. Note the use of IC in the Case Control command versus TIC on the Bulk Data entry. For heat transfer, the IC Case Control command selects TEMP or TEMPD entries for initial conditions and not the TIC entry.
2. If no TIC set is selected in the Case Control Section, all initial conditions are assumed to be zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed to be zero.
4. In direct transient analysis (SOL 109 and 129) as well as in modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC or IC(PHYSICAL) Case Control command, G may reference only grid, scalar or extra points. In modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC(MODAL) Case Control command, G may reference only modal coordinates or extra points.
5. The initial conditions for the independent degrees-of-freedom specified by this Bulk Data entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and/or TLOAD2 Bulk Data entries.
6. SOL700 does not support U0 (initial displacement).

**TICEL****Transient Initial Conditions of Elements**

Defines the initial values of element variables at the beginning of the analysis. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3     | 4     | 5      | 6     | 7      | 8      | 9 | 10 |
|-------|-----|-------|-------|--------|-------|--------|--------|---|----|
| TICEL | SID | SETID | NAME1 | VALUE1 | NAME2 | VALUE2 | -etc.- |   |    |

**Example:**

|       |   |    |         |      |     |      |  |  |  |
|-------|---|----|---------|------|-----|------|--|--|--|
| TICEL | 3 | 40 | DENSITY | 100. | SIE | 1.E5 |  |  |  |
|-------|---|----|---------|------|-----|------|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| SID        | Set identification number. (Integer > 0; Required)                                      |
| SETID      | Number of a SET1 entry defining the elements to be initialized. (Integer > 0; Required) |
| NAMEi      | Element variable to be initialized. See Remark 5. (Character; Required)                 |
| VALUEi     | Value of the variable. (Real; Required)   |

**Remarks:**

- Initial conditions for elements that are not specified on TICEL entries are assumed to be zero except density, which is set to the reference density.
- Only initial conditions that are selected in the Case Control Section (IC = SID) will be activated by the solver.
- As many continuation lines as required can be used to specify all the variables being initialized. A blank field terminates the list.
- Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder,...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence, and overrules the TICEUL1 initialization for the element.
- The following variables for NAMEi can be used to initialize the Eulerian regions:

|         |                          |
|---------|--------------------------|
| XVEL    | x-velocity               |
| YVEL    | y-velocity               |
| ZVEL    | z-velocity               |
| DENSITY | Density                  |
| SIE     | Specific internal energy |
| Q       | Artificial viscosity     |

|      |                   |
|------|-------------------|
| DIV  | Divergence        |
| VOID | Void fraction     |
| FMAT | Material fraction |
| XMOM | x-momentum        |
| YMOM | y-momentum        |
| ZMOM | z-momentum        |

6. To initialize the pressure use density. And depending on the equation of states also define the specific internal energy (SIE).
7. For the Euler solvers, you can, in addition to the “normal” element variables that the solver has defined, also define an initial radial velocity field. You have to enter the location of the center from where the radial emerges, the velocity to be applied to the element center and the decay coefficient for the velocity field. The center is defined by the keywords “X-CENTER, Y-CENTER, Z-CENTER”, the radial velocity by “R-VEL” and the decay coefficient by "DECAY". You have to input these keywords in the above order, and have every keyword followed by its value.

## TICEUL1

### Transient Initial Conditions of Eulerian Regions

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

Format:

| 1       | 2   | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|------|---|---|---|---|---|---|----|
| TICEUL1 | SID | TSID |   |   |   |   |   |   |    |

Example:

|         |     |     |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|
| TICEUL1 | 300 | 200 |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Unique TICEUL1 number referenced from a PEULER1 entry. (Integer > 0; Required) |
| TSID       | Group of geometric region TICREG ID. (Integer > 0; Required)                   |

Remarks:

1. Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.

**TICEUDS**

User-defined Transient Initial Conditions of Euler Elements or Lagrangian grid points.

Defines the initial values of element or grid point variables at the beginning of the analysis by a user written subroutine. Use in SOL700 only.

**Format:**

| 1       | 2   | 3     | 4     | 5     | 6    | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|-------|------|---|---|---|----|
| TICEUDS | SID | GROUP | UNAME | SETID | COPT |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE initex 'SCAI.MDSolver.Obj.Uds.Dytran.InitOut'
```

*In Bulk Data:*

| 1       | 2  | 3      | 4      | 5   | 6    | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|-----|------|---|---|---|----|
| TICEUDS | 12 | initex | EXINIT | 100 | GRID |   |   |   |    |

| Descriptor | Meaning  |   |
|------------|--|---|
| SID        | Unique output number. (Integer > 0; Required)  |   |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)       |   |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXINIT)                      |   |
| SETID      | Number of a SET1 entry defining the elements or grids to be initialized. (Integer > 0, Required) |   |
| COPT       | Flag for assigning initial velocities. (Character; default=Element)                              |   |
|            | Element  | Apply the initial variables on elements |
|            | GRID   | Apply the initial variables on grids    |

**Remarks:**

1. Only can be used for SOL 700.
2. Initial conditions to be used must be selected in the Case Control Section (TIC = SID).
3. UNAME=EXINIT can only be used.

**TICREG****Transient Initial Conditions of Eulerian Regions**

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

**Format:**

| 1      | 2    | 3    | 4    | 5   | 6   | 7    | 8     | 9 | 10 |
|--------|------|------|------|-----|-----|------|-------|---|----|
| TICREG | TRID | TSID | TYPE | VID | MID | TSID | LEVEL |   |    |

**Example:**

|        |     |     |        |     |     |   |     |  |  |
|--------|-----|-----|--------|-----|-----|---|-----|--|--|
| TICREG | 300 | 200 | SPHERE | 400 | 100 | 3 | 4.0 |  |  |
|--------|-----|-----|--------|-----|-----|---|-----|--|--|

| Descriptor | Meaning  |   |
|------------|--|---|
| TRID       | Unique TRID number. (Integer > 0; Required)  |   |
| TSID       | ID of group of Euler regions referenced from the TICEUL1 entry. (Integer > 0; Required)                      |   |
| TYPE       | The type of Eulerian region. (Character; Required)   |   |
|            | SURF   | Region inside or outside a surface.       |
|            | SPHERE   | Region inside a sphere.                   |
|            | CYLINDER   | Region inside a cylinder.                 |
|            | BOX  | Region inside a box.                      |
|            | ELEM   | Region defined by list of Euler elements. |
| VID        | ID of a geometric entity. (Integer > 0; Required)  |   |
|            | Type:  | Region:                                   |
|            | SURF   | SURFINI                                   |
|            | SPHERE   | SPHERE                                    |
|            | CYLINDER   | CYLINDR                                   |
|            | BOX  | BCBOX                                     |
|            | ELEM   | SET1                                      |
| MID        | ID of a MATDEUL entry defining this material. (Integer $\geq 0$ ; Default = 0)                               |   |
| TSID       | ID of a TICVAL entry containing a list of initial values for this material. (Integer $\geq 0$ ; Default = 0) |   |
| LEVEL      | Level indicator for this material and initial values. (Real; Default = 0.0)                                  |   |

**Remarks:**

1. A number of TICREG may exist in the input file with the same TSID. The TICEUL entry will combine all TICREGs with the same TSID into one initial definition for the Euler elements that are referenced from the same PEULER1 definition.
2. When the material number is left blank or zero, the Eulerian elements inside the region will be void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1stOrder or 2ndOrder scheme, an error message will be issued and the analysis will stop.
3. All level indicators LEVEL of the same TSID group must have different values. The level indicator can be negative.
4. See also the parameter MICRO for the accuracy of the initial value generation.

**TICVAL****Transient Initial Condition Set**

Defines the initial values of an Eulerian geometric region. Used in SOL 700 only.

**Format:**

| 1      | 2     | 3      | 4      | 5      | 6     | 7      | 8     | 9      | 10 |
|--------|-------|--------|--------|--------|-------|--------|-------|--------|----|
| TICVAL | TSID  | METHOD | NAME1  | VALUE1 | NAME2 | VALUE2 | NAME3 | VALUE3 |    |
|        | NAMEi | VALUEi | -etc.- |        |       |        |       |        |    |

**Example:**

|        |      |     |         |      |      |     |     |     |  |
|--------|------|-----|---------|------|------|-----|-----|-----|--|
| TICVAL | 3    |     | DENSITY | 100. | YVEL | 25. | SIE | 3.7 |  |
|        | XVEL | 3.5 |         |      |      |     |     |     |  |

| Descriptor | Meaning  |
|------------|--|
| TSID       | Unique TICVAL number referenced from a TICEUL entry. (Integer > 0; Required)   |
| METHOD     | Type of input definition. (Character; Default = NORMAL)  |
| RADIAL     | Initializes material with radial profiles. The entries VALUEi are interpreted as TABLED1 IDs. See Remarks 2., 3., and 4. |
| NORMAL     | Normal initialization.   |
| NAMEi      | Variable to be initialized. See <a href="#">TICEL, 3251</a> . (Character; Required)                                      |
| VALUEi     | Value of the variable. (Real; Required)  |

**Remarks:**

- Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When an Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.
- METHOD = RADIAL allows to map results of a spherical symmetric 1-D solution onto a full 3-D model. For initialized variables SIE and DENSITY, a 1-D table has to be defined that specifies the variable value for a number of distances from the center. The center is by default (0,0,0) but can be changed by setting X-CENTER, Y-CENTER, Z-CENTER. The velocity is a radial velocity and has to be specified as R-VEL. Its values is also a TABLED1 ID.
- DYPARAM,SPHERSYM can be used to define a proper 1-D spherical mesh and speeds up the run by taking only the mesh-size in radial direction into account.

4. Radial initialization of JWL is supported. The entries DETSPH and the JWL entry from the 1-D spherical solution stage have to be included in the remap run. Alternatively, the 1-D solution may be run with JWL and the follow-up run with ideal gas, provided that all JWL material has fully ignited. Radial initialization of EOSIG is not supported. In the follow-up run, ideal gas material has to be used instead of IG material.

**TIC3****Transient Analysis Initial Velocity with Increment Options**

Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in SOL 700 only.

**Format:**

| 1    | 2    | 3    | 4    | 5     | 6    | 7    | 8 | 9 | 10 |
|------|------|------|------|-------|------|------|---|---|----|
| TIC3 | SID  | G    |      | SCALE |      |      |   |   | +  |
| +    | XVEL | YVEL | ZVEL | XROT  | YROT | ZROT |   |   | +  |
| +    | G1   | G2   | THRU | G3    | BY   | G4   |   |   |    |

**Example:**

|      |       |   |      |      |    |      |  |  |   |
|------|-------|---|------|------|----|------|--|--|---|
| TIC3 | 7     | 5 |      | 10.  |    |      |  |  | + |
| +    | 100.0 |   |      | 5.0  |    | -7.5 |  |  | + |
| +    | 1     | 2 | THRY | 1000 | BY | 23   |  |  |   |

| Descriptor          | Meaning   |
|---------------------|---|
| SID                 | Number of a set of loads. (Integer > 0; Required)   |
| G                   | Number of a grid point at the center of rotation. (Integer > 0; Required)   |
| SCALE               | Scale factor of initial velocity. (Real, Default=1.0)   |
| XVEL, YVEL,<br>ZVEL | Initial translational velocity components. (Real; Default = 0.0)  |
| XROT, YROT,<br>ZROT | Initial rotational velocity components. (Real; Default = 0.0)   |
| G1, G2, ...         | Grid points to be initialized. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer > 0; Required) |

**Remarks:**

1. Any number of TIC3 entries can be used.
2. The rotational velocity components are defined in radians per unit time.
3. For six degree of freedom grid points, the angular velocity components are also initialized.
4. Initial conditions for grid points that are not specified on TICn entries are assumed to be zero.
5. If the THRU specification is used, the grid points in the range definition are required to exist. The BY option enables grid points to be ignored in this range.
6. None of the fields in the list of grid points can be blank or zero, since this indicates the end of the list.
7. The initial conditions to be used in SOL 700 must be selected in the Case Control Section (IC = SID).

8. If grid points are part of a rigid body, it is recommended you enable double precision in SOL700. It is possible that a single precision SOL700 analysis will not assign the correct initial velocities.

**TIM2PSD**

## FFT conversion tool for use in random vibration fatigue analysis

Fast Fourier Transformation (FFT) conversion tool for use in SOL 108 or SOL 111 random vibration fatigue analysis. This entry is used to convert time history data into power spectral density (PSD) functions directly input to the fatigue analysis.

**Format:**

| 1       | 2        | 3       | 4      | 5      | 6      | 7     | 8     | 9          | 10 |
|---------|----------|---------|--------|--------|--------|-------|-------|------------|----|
| TIM2PSD | ID       | SRATE   | WINDOW | FORMAT | MEANS  | NSKIP |       |            |    |
|         | “EVENT”  | EVNTID1 |        |        |        |       | T1    | $\delta 1$ |    |
|         |          | EVNTID2 |        |        |        |       | T2    | $\delta 2$ |    |
|         |          | ...     |        |        |        |       |       |            |    |
|         |          | EVNTIDn |        |        |        |       | Tn    | $\delta n$ |    |
|         | “DELETE” | EVNTID1 | Ti1_1  | Tf1_1  | Ti2_1  | Tf2_1 | Ti3_1 | Tf3_1      |    |
|         |          | EVNTID2 | Ti1_2  | Tf1_2  | Ti2_2  | Tf2_2 | Ti3_2 | Tf3_2      |    |
|         |          | ...     |        |        |        |       |       |            |    |
|         |          | EVNTIDn | Ti1_n  | Tf1_n  | Ti2_n  | Tf2_n | Ti3_n | Tf3_n      |    |
|         | “MAP”    | LCID1   | CHAN1  | LCID2  | CHAN2  | LCID3 | CHAN3 |            |    |
|         |          | LCID4   | CHAN4  | ...    | -etc.- |       |       |            |    |

**Example:**

|         |        |       |             |     |     |   |     |     |  |
|---------|--------|-------|-------------|-----|-----|---|-----|-----|--|
| TIM2PSD | 42     | 512.0 | HANNIN<br>G | CSV | YES | 1 |     |     |  |
|         | EVENT  | 80001 |             |     |     |   | 2.0 | 1.0 |  |
|         |        | 80002 |             |     |     |   | 1.0 | 0.5 |  |
|         |        | 80003 |             |     |     |   | 2.0 | 1.0 |  |
|         | DELETE | 80001 | 1.1         | 1.9 |     |   |     |     |  |
|         | MAP    | 1     | 3           | 2   | 5   | 3 | 7   |     |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique identification number referenced by FATIGUE case control, otherwise TIM2PSD entry is ignored (Integer > 0).  |
| SRATE      | Number of samples per second (required) - thus dt = 1/SRATE (Real; No Default).   |
| WINDOW     | Window function to use. Choices are HANNING or NONE. This is applied to the "block" of data extracted from the total time signal. (Character; Default = HANNING). |

| Descriptor                                | Meaning  |
|---|--|
| FORMAT                                    | Format of time signal files (RPC or CSV) (Character; Default = CSV). RPC files are binary channel data usually with the .rsp file extension. CSV files are typical MS Excel comma separated files in which the channels are in the columns (i.e. three columns of data is three channels of data - y-values only. x-values assumed to be in ascending order with no regards to length of time between points). |
| MEANS                                     | Used to decide if overall signal mean values are to be calculated (YES or NO) (Character; Default = NO). Currently this is only calculated and not applied in the analysis. Ignored if no mean stress correction specified on FTGPARM entry.   |
| NSKIP                                     | Number of header lines to skip if an FORMAT=CSV file (Integer>=0; Default=0).<br><br>The next continuation lines are required and repeat for as many Events as necessary.  |
| EVENT                                     | Flag indicating that event parameters are to follow (one optional set for each event).   |
| EVNTID <sub>i</sub>                       | Number of this Event - must correspond to the TID of a FTGLOAD entry of TYPE=PSD that is referenced by an active FTGEVNT entry. (Note: it is NOT the actual FTGEVNT ID).   |
| T <sub>i</sub>                            | Length of window function in time for this Event (Real>0.0; no Default).   |
| δ <sub>i</sub>                            | Overlap or gap in time between windows for this Event (real) (+ means overlap) (Real>=0.0; Default = 0.0).<br><br>The next continuation lines are optional and can repeat for as many Events as necessary.   |
| DELETE                                    | Flag indicating that event parameters are to follow for signal deletion. For any Event, time value pairs can be specified for deleting up to three (3) portions of the time signal for each Event.   |
| EVNTID <sub>i</sub>                       | Number of this Event - correspond to a previously defined EVNTID <sub>i</sub> in the "EVENT" section above.  |
| T <sub>i</sub> <sub>1</sub> <sub>_i</sub> | Used to specify sections in an Event (defined by pairs of time values t <sub>1</sub> -t <sub>2</sub> , t <sub>3</sub> -t <sub>4</sub> , t <sub>5</sub> -t <sub>6</sub> )   |
| T <sub>f</sub> <sub>1</sub> <sub>_i</sub> | to delete from Event files before FFT process is applied (Real>0; t <sub>1</sub> <=t <sub>2</sub> <=t <sub>3</sub> <t <sub>4</sub> <=...; no Defaults; if none are specified, entire signal is used). Only three (3) delete pairs per event are currently supported.<br><br>The next continuation lines are optional and only necessary if the SUBCASEs corresponding to the channel data is not one-to-one.   |
| MAP                                       | Flag used to map the channel data and load case (SUBCASE) IDs.   |
| LCID <sub>i</sub>                         | SUBCASE of transfer function (TF) corresponding to <i>i</i> th load event. SUBCASEs must be in ascending order in the Case Control! There cannot be more SUBCASEs listed than there are channels of data in the CSV/RPC files.   |
| CHAN <sub>i</sub>                         | Channel # corresponding to <i>i</i> th load event. Channels must be referenced in ascending order and cannot be repeated.  |

**Remarks:**

1. TIM2PSD bulk data entries are ignored if not selected by a FATIGUE case control.

2. The actual RPC/CSV files are specified with a UDNAME entry referenced by TID field on the FTGLOAD entry for each Event (FTGEVNT). This is a requirement in the presence of a TIM2PSD entry.
3. All events must be in the same format, use the same window function, have the same sample rate, and have the same number and order of channels.
4. Each event can be a different length and can have a different block length and gap.
5. If LOGLVL=1 or 2 on the FTGPARM entry, the input time histories and direct PSD computations are written to CSV files for possible plotting with MS Excel.
6. The TID on the FTGLOAD is also used as the SID of the auto-generated RANDPS entries in this process.
7. A file containing the auto-generated RANDPS and TABRND1 entries is created in the same directory as the referenced UDNAME filename. This file can then be used as an include file for subsequent analyses that use the same generated PSD data rather than using the TIM2PSD entry again in subsequent runs.
8. This process also determines the overall mean values of the time histories for each event. The means are normally thrown away in conversion to PSDs and cross-PSDs. Currently the mean effect is not taken into account in the subsequent fatigue analysis. These means are appended to the end of the file containing the auto-generated RANDPS and TABRND1 entries.
9. MAP flag is not necessary if all channels in RPC/CSV file are used in the exact same order as the SUBCASEs corresponding to the transfer functions. SUBCASE IDs must be in ascending order. Channels must be referenced in ascending order.

**TLOAD1**

## Transient Response Dynamic Excitation, Form 1

Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A\} \cdot F(t - \tau)$$

for use in transient response analysis.

**Format:**

| 1      | 2   | 3        | 4                 | 5    | 6     | 7   | 8   | 9 | 10 |
|--------|-----|----------|-------------------|------|-------|-----|-----|---|----|
| TLOAD1 | SID | EXCITEID | DELAYI/<br>DELAYR | TYPE | TID/F | US0 | VS0 |   |    |

**Example:**

|        |   |   |    |      |    |  |  |  |  |
|--------|---|---|----|------|----|--|--|--|--|
| TLOAD1 | 5 | 7 | 15 | LOAD | 13 |  |  |  |  |
|--------|---|---|----|------|----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0) See Remarks 1. and 5.   |
| EXCITEID   | Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$ . See Remarks 2. and 3. (Integer > 0) |
| DELAYI     | Identification number of DELAY Bulk Data entry that defines time delay $\tau$ . See Remark 8. and 13. (Integer > 0 or blank)   |
| DELAYR     | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 8. and 13. (Real or blank)                           |
| TYPE       | Defines the type of the dynamic excitation. See Remarks 2., 3. and 12. (Integer, character or blank; Default = 0)  |
| TID/F      | Identification number of TABLEDi entry that gives $F(t)$ . (Integer > 0). Value of F to be used for all times. (Real, non-zero)  |
| US0        | Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9., 11. and 14. (Real; Default = 0.0)   |
| VS0        | Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10., 11. and 14. (Real; Default = 0.0)   |

**Remarks:**

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

| TYPE                   | TYPE of Dynamic Excitation   |
|------------------------|--|
| 0, L, LO, LOA or LOAD  | Applied load (force or moment) (Default)   |
| 1, D, DI, DIS, or DISP | Enforced displacement using large mass or SPC/SPCD data                          |
| 2, V, VE, VEL or VELO  | Enforced velocity using large mass or SPC/SPCD data.                             |
| 3, A, AC, ACC or ACCE  | Enforced acceleration using large mass or SPC/SPCD data                          |
| 4                      | FLOW boundary condition on the face of an Eulerian solid element (SOL 700 only). |
| 12                     | Velocity of the center of gravity of a rigid body (SOL 700 only)                 |
| 13                     | Force or moment on the center of gravity of a rigid body (SOL 700 only).         |

For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400.

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, static and thermal load set entries

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

4. EXCITEID may reference sets containing QHBDY, QBDYi, QVECT, QVOL and TEMPBC entries when using the heat transfer option.
5. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
6. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
7. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. Please refer to [Fourier Transform](#) (Ch. 20) in the *MSC Nastran Dynamic Analysis User's Guide*.
8. If the DELAYI/DELAYR field is blank or zero,  $\tau$  will be zero.
9. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(US0)$  where  $\{A\}$  is defined by the EXCITEID field.
10. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(VS0)$  where  $\{A\}$  is defined by the EXCITEID field.
11. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
12. For TYPE=4, TID must be blank if it references a FLOW entry. Use the FLOWT entry to define a time dependent flow boundary on the face of an Eulerian element.
13. For RC network solver in thermal analysis, the DELAY1/DELAYR is ignored.
14. Fields US0 and VS0 are not supported in SOL 700.
15. DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.
16. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TLOAD2****Transient Response Dynamic Excitation, Form 2**

Defines a time-dependent dynamic excitation or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\}\tilde{t}^B e^{\tilde{C}\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \quad (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

for use in a transient response problem, where  $\tilde{t} = t - T1 - \tau$

Format:

| 1      | 2   | 3        | 4                 | 5    | 6  | 7  | 8 | 9 | 10 |
|--------|-----|----------|-------------------|------|----|----|---|---|----|
| TLOAD2 | SID | EXCITEID | DELAYI/<br>DELAYR | TYPE | T1 | T2 | F | P |    |
|        | C   | B        | US0               | VS0  |    |    |   |   |    |

Example:

|        |     |    |     |  |     |     |      |  |  |
|--------|-----|----|-----|--|-----|-----|------|--|--|
| TLOAD2 | 4   | 10 | 5.0 |  | 2.1 | 4.7 | 12.0 |  |  |
|        | 2.0 |    |     |  |     |     |      |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. See Remarks 1. and 6. (Integer > 0)   |
| EXCITEID   | Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$ . See Remarks 2. and 3. (Integer > 0) |
| DELAYI     | Identification number of DELAY Bulk Data entry that defines time delay $\tau$ . See Remark 5. (Integer > 0 or blank)   |
| DELAYR     | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)                                   |
| TYPE       | Defines the type of the dynamic excitation. See Remarks 2. and 3. (Integer; character or blank; Default = 0)   |
| T1         | Time constant. (Real $\geq 0.0$ )  |
| T2         | Time constant. (Real; $T2 > T1$ )  |
| F          | Frequency in cycles per unit time. (Real $\geq 0.0$ ; Default = 0.0)   |
| P          | Phase angle in degrees. (Real; Default = 0.0)  |
| C          | Exponential coefficient. (Real; Default = 0.0)   |
| B          | Growth coefficient. (Real; Default = 0.0)  |

| Descriptor | Meaning  |
|------------|--|
| US0        | Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9., 11. and 16. (Real; Default = 0.0) |
| VSO        | Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10., 11. and 16.(Real; Default = 0.0)    |

**Remarks:**

1. Dynamic excitation sets must be selected with the Case Control command with DLOAD=SID.
2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

| TYPE                   | TYPE of Dynamic Excitation   |
|------------------------|--|
| 0, L, LO, LOA or LOAD  | Applied load (force or moment) (Default)                                 |
| 1, D, DI, DIS, or DISP | Enforced displacement using large mass or SPC/SPCD data                  |
| 2, V, VE, VEL or VELO  | Enforced velocity using large mass or SPC/SPCD data                      |
| 3, A, AC, ACC or ACCE  | Enforced acceleration using large mass or SPC/SPCD data                  |
| 12                     | Velocity of the center of gravity of a rigid body. (SOL 700 only)        |
| 13                     | Force or moment on the center of gravity of a rigid body. (SOL 700 only) |

For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400.

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control

EXCITEID may reference DAREA, static and thermal load set entries

- There is a LOADSET request in Case Control

The program may reference DAREA entries as well as static and thermal load set entries specified by the LID or TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control

EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

- EXCITEID (field 3) may reference sets containing QHBDY, QBDYi, QVECT, and QVOL and TEMPBC entries when using the heat transfer option.
- If the DELAYI/DELAYR field is blank or zero,  $\tau$  will be zero.
- SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
- If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
- If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. In this case, B will be rounded to the nearest integer. Please refer to [Fourier Transform](#) (Ch. 20) in the *MSC Nastran Dynamic Analysis User's Guide*.
- The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(US0)$  where  $\{A\}$  is defined by the EXCITEID field.
- The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(VS0)$  where  $\{A\}$  is defined by the EXCITEID field.
- The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
- The continuation entry is optional.
- TYPE=4 (SOL 700) is not supported using TLOAD2.
- For SOL 700, TLOAD2 is converted to TLOAD1 and a TABLED1 is generated internally.
- RC network solver does not support TLOAD2 for thermal analysis.
- Fields US0 and VS0 are not supported in SOL 700.
- DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.

18. If comparing results between pre-2005 versions and later versions, please note that results are shifted by one time step in later versions. The tload2 should be updated accordingly if there is a need to compare to these pre-2005 versions.
19. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TMPSET****Temperature Group Set Definition**

Define a time-dependent dynamic thermal load group for use in TTEMP Bulk Data entry in SOL 400.

**Format:**

| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|----|----|----|----|----|----|----|----|----|
| TMPSET | ID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

**Alternate Format:**

|        |    |    |        |    |      |     |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|
| TMPSET | ID | G1 | "THRU" | G2 | "BY" | INC |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

**Continuation Entry Format 1:**

|  |    |    |     |     |        |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|
|  | G8 | G9 | G10 | G11 | -etc.- |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|

**Continuation Entry Format 2:**

|  |    |        |    |      |     |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|
|  | G8 | "THRU" | G9 | "BY" | INC |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|

**Example:**

|        |    |      |      |    |    |    |  |  |  |
|--------|----|------|------|----|----|----|--|--|--|
| TMPSET | 15 | 5    | THRU | 21 | BY | 4  |  |  |  |
|        | 27 | 30   | 32   | 33 |    |    |  |  |  |
|        | 35 | THRU | 44   |    |    |    |  |  |  |
|        | 67 | 68   | 72   | 75 | 84 | 93 |  |  |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Temperature group identification number. (Integer > 0)        |
| Gi         | Grid point Identification numbers in the group. (Integer > 0) |

**Remarks:**

1. This entry is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0).
2. GROUP\_ID determines the group of a specified the time-dependent distribution of temperatures. It is used by the TTEMP Bulk Data entry to define the corresponding TABLEDi entry. GROUP\_ID must be unique for all of the other TMPSET entries.
3. The temperature of grid point Gi must be defined using TEMP, TEMPD, TEMPP1, or TEMPRB Bulk Data entry. These bulk data entries must have the same SID as that referenced on the associated TTEMP Bulk Data entry.

**TODYNA**

Defines the Start of Direct Text to Dytran

All bulk data entries between TODYNA and ENDDYNA will be passed directly from SOL 700 to Dytran.  
Used in SOL 700 only.

Format:

| 1      | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---|---|---|---|---|---|---|---|----|
| TODYNA |   |   |   |   |   |   |   |   |    |

Example:

|        |     |        |  |       |        |  |  |  |  |
|--------|-----|--------|--|-------|--------|--|--|--|--|
| TODYNA |     |        |  |       |        |  |  |  |  |
| MAT1   | 345 | 29.0E6 |  | 0.285 | 0.0004 |  |  |  |  |
| ENDDYN |     |        |  |       |        |  |  |  |  |

**TOMVAR****Topometry Design Variable**

Defines a design region for topometry optimization (element-by-element optimization).

**Format:**

| 1      | 2        | 3     | 4   | 5         | 6     | 7   | 8   | 9     | 10 |
|--------|----------|-------|-----|-----------|-------|-----|-----|-------|----|
| TOMVAR | ID       | TYPE  | PID | PNAME/FID | XINIT | XLB | XUB | DELXV |    |
|        | “DLINK”  | TID   | C0  | C1        |       |     |     |       |    |
|        | “DDVAL”  | DSVID |     |           |       |     |     |       |    |
|        | “STRESS” | STLIM |     |           |       |     |     |       |    |

**Example:**

Design all element's thickness referencing PSHELL ID = 5 with initial design = 10.0 ( $t_0 = 10.0$  input element thickness), lower bound  $0.5 \cdot t_0$  and upper bound  $1.5 \cdot t_0$ .

|        |        |     |        |   |   |      |  |  |  |
|--------|--------|-----|--------|---|---|------|--|--|--|
| TOMVAR | 1      | PS1 | PSHELL | 5 | T | 10.0 |  |  |  |
|        | STRESS | 5.0 |        |   |   |      |  |  |  |

**Example:**

Design all element's Young Modulus referred by PSHELL ID = 100 with initial design XINIT = 3.E+5, XLB=1.0, and XUB= 1.0E+6.

|        |    |        |     |   |       |     |       |  |  |
|--------|----|--------|-----|---|-------|-----|-------|--|--|
| TOMVAR | 10 | PSHELL | 100 | E | 3.E+5 | 1.0 | 1.E+6 |  |  |
|--------|----|--------|-----|---|-------|-----|-------|--|--|

| Descriptor | Meaning  |
|------------|--|
| ID         | Unique topometry design region identification number. (Integer > 0)  |
| TYPE       | Property entry type. Used with PID to identify the elements to be designed. See Remark 2. (Character: “PBAR”, “PSHELL”, ‘PSOLID’, and “PCOMP”, etc.)   |
| PID        | Property entry identifier (Integer > 0). This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1, DVPREL2, DVMREL1, and DVMREL2 entries. Topometry, topology, and sizing variables cannot share the same properties. (Integer > 0). Combined topometry, topology, topography, sizing, and shape variables are allowed.  |
| PNAME/FID  | Property name or property material name, such as “T”, “A”, “E”, and “GE”, or field position of the property entry or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referenced by field position. Only one property value for each property can be designed by TOMVAR in a job. See Remark 2. (Character or Integer > 0.) |

| Descriptor | Meaning  |
|------------|--|
| XINIT      | Initial value. (Real or blank, no Default). Typically, XINIT is defined to match the mass target constraint (so the initial design does not have violated constraints) or the analysis model input property value. |
| XLB        | Lower bound. (Real or blank; Default = blank) . The default is XLB=0.5*XINIT.  |
| XUB        | Upper bound . (Real or blank; Default = blank). The default is XLB=1.5*XINIT.  |
| DELXV      | Fractional change allowed for the design variable during approximate optimization. See Remark 6. (Real > 0.0; Default = 0.5)   |
| “DLINK”    | Indicates that this line relates a ply thickness to another ply thickness. See Remark 10.  |
| TID        | TOMVAR entry identifier. (Integer > 0)   |
| C0         | Constant term. (Real; Default = 0.0)   |
| C1         | Coefficient term. (Real; no Default)   |
| “DDVAL”    | Indicates that this line defines discrete TOMVAR variables.  |
| DSVID      | DDVAL entry identifier. (Integer > 0)  |
| “STRESS”   | Indicates that this line defines a stress limit.   |
| STLIM      | Von Mises stress upper bound. See Remark 11. (Real >0.0)   |

### Remarks:

1. Multiple TOMVAR's are allowed in a single file.
2. Property name and FID > 0 can be used for element property values just like the DVPREL1 Bulk Data entry. Only property name can be used for material property values like DVMREL1.
3. Property name “A” is shared by material and property entries. If “A” is used for PNAME, it is a Material. For PROD, PBEAM and PBAR, the integer field ID must be used to point to the area of these entries. TOMVAR supports MAT1 only.
4. PBARL, PBEAML, PBRSECT and PBMSECT are not supported.
5. For TYPE = PSHELL and PNAME selecting a material property, all the MATi fields on the PSHELL must be the same.
6. Combined topometry, topography, topology, sizing, and shape optimization is supported in a single file. However, topometry and topology cannot reference the same property ID. It is possible to topometry certain elements while sizing others. It is allowed to simultaneously design the same elements with topometry and desvar (sizing and/or shape) variables but topometry and sizing cannot reference the same property name.
7. Topometry optimization with element response constraints is slow due to many design variables. In this case, fully stressed design (FSD) is an alternative for certain problems
8. Parameters DESPCH and DESPCH1 specify when the topometry optimized design values are written to the element result history file jobname.des that can be imported to Patran and other post-processor to view topometry optimized results.
9. The TOMVAR entry cannot be used with thermal loads.

10. The DLINK line can only be used when TYPE=PCOMP. The PID must be the same as the PID given on the TOMVAR referenced by TID and the PNAMES in the two TOMVAR entries must differ
11. “STRESS” limits can only be used for PTYPE=PSHELL and PNAME= T. The Von Mises stress (at element center) constraints apply to all shell elements in both designed and non-designed regions. All TOMVAR entries must have the same STLIM.

**TOPVAR**

## Topological Design Variable

Define a topology design region for topology optimization.

Format:

| 1      | 2        | 3     | 4     | 5     | 6     | 7     | 8     | 9   | 10 |
|--------|----------|-------|-------|-------|-------|-------|-------|-----|----|
| TOPVAR | ID       | LABEL | PTYPE | XINIT | XLB   | DELXV | POWER | PID |    |
|        | “SYM”    | CID   | MS1   | MS2   | MS3   | CS    | NCS   |     |    |
|        | “CAST”   | CID   | DD    | DIE   | ALIGN |       |       |     |    |
|        | “EXT”    | CID   | ED    | ALIGN |       |       |       |     |    |
|        | “PRINT”  | CID   | PD    |       |       |       |       |     |    |
|        | “TDMIN”  | TVMIN | TVMAX |       |       |       |       |     |    |
|        | “STRESS” | STLIM |       |       |       |       |       |     |    |

Example:

|        |        |     |        |   |   |      |   |   |  |
|--------|--------|-----|--------|---|---|------|---|---|--|
| TOPVAR | 1      | PS1 | PSHELL | 5 | T | 10.0 | 4 | 1 |  |
|        | STRESS | 5.0 |        |   |   |      |   |   |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Unique topology design region identification number. (Integer > 0)  |
| LABEL      | User-supplied name for printing purpose. (Character)  |
| PTYPE      | Property entry name. Used with PID to identify the elements to be designed. (Character: “PBAR”, “PSHELL”, ‘PSOLID’, etc.)   |
| XINIT      | Initial value. (Blank or Real, XLB < XINIT ≤ 1.0; Default = Blank). Typically, XINIT is defined to match the mass constraint on DRESP1=FRMASS, so the initial design does not have violated constraints. In this case, the default is set to the constraint value. If the mass (DRESP1=FRMASS or WEIGHT) is the objective, the default is 0.9. The default of XINIT is 0.6 for the other cases. |
| XLB        | Lower bound to prevent the singularity of the stiffness matrix. (Real; Default = 0.001)   |
| DELXV      | Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)  |
| POWER      | A penalty factor used in the relation between topology design variables and element Young’s modulus. (Real > 1.0; Default = 3.0, or 4.0 if symmetry constraints used). 2.0 ≤ POWER ≤ 5.0 is recommended.  |
| PID        | Property entry identifier. This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1 and DVPREL2 entries. Topology and sizing variables cannot share the same properties. (Integer > 0)  |
| “SYM”      | Indicates that this line defines symmetry constraints.  |

| Descriptor | Meaning  |
|------------|--|
| CID        | Rectangular coordinate system ID used for specifying manufacturing constraints. See Remark 4. (Blank or Integer > 0; Default = blank)  |
| MSi        | Mirror symmetry plane. See Remark 5. & 7. (Character, 'XY', 'YZ', or 'ZX')   |
| CS         | Cyclic symmetry axis. (character X, Y, Z). See Remark 12.  |
| NCS        | Number of cyclic symmetric segments in 360 degrees (Integer > 0). See Remark 9.  |
| "CAST"     | Indicates that this line defines casting constraints (i.e., die draw direction constraints). See Remarks 6., 7., 8., and 10.   |
| DD         | Draw Direction. DDi=X, Y, Z or X-, Y-, Z- for a single die option (DIE=1) where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively. DDi=X, Y, and Z for two die option (DIE =2) (Character)   |
| DIE        | Die Options. (Blank or integer 1 or 2; Default = 1)<br>= 1 (or blank). A single die will be used and the die slides in the given draw direction (i.e., material grows from the bottom in the draw direction)<br>= 2. Two dies will be used and the dies split apart along the draw direction (i.e., material grows from the splitting plane in opposite direction along the axis specified by the draw direction DDi. The splitting plane is determined by optimization) |
| ALIGN      | Indicates whether the designed property finite element mesh is precisely aligned with the draw direction or extrusion direction. (Character: "YES" or "NO" or Blank; Default = blank = "NO") See Remark 10.  |
| "EXT"      | Indicates that this line defines extrusion constraints (i.e., enforce constant cross-section) See Remark 6. and 7.   |
| ED         | Extrusion direction. (Character, X, Y, or Z)   |
| "PRINT"    | Indicates that this line defines overhang constraints (maximum overhang angle is 45 degree). See Remark 12.  |
| CID        | Rectangular coordinate system ID used for specifying overhang constraints.   |
| PD         | PRINT Direction. PD =X, Y, Z or X-, Y-, and Z- where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively (Character).  |
| "TDMIN"    | Indicates that this line defines a minimum and/or maximum member size., See remarks 11. and 12.  |
| TVMIN      | Minimum member size. See Remarks 11. and 12. (Real>=0.0 or blank)  |
| TVMAX      | Maximum member size. See Remarks 11. and 12. (Real > TVMIN or blank)   |
| "STRESS"   | Indicates that this line defines a stress limit.   |
| STLIM      | Von Mises stress upper bound. See Remark 13.. (Real >0.0)  |

**Remarks:**

1. The topologically designable element properties include PROD, PBAR, PBARL, PBEND, PBEAM, PBEAML, PSHELL, PSHEAR, PSOLID, and PWELD. Multiple TOPVAR's are allowed in a single file. Combined topology, topography (BEADVAR), topometry (TOMVAR) sizing, and shape optimization is supported in a single file. However, TOPVAR cannot be used with DVMREL1 and DVMREL2 entries.
2. All designed element properties must refer to a MAT1 entry or MAT9 entry (PSOLID only); therefore, a PCOMP/PCOMPG cannot be used as designed property in topology optimization. PCOMP/PCOMPG's can be used as non-designed properties in a topology optimization job.
3. If DELXV is blank, the default is taken from the specification of DELX parameter on the DOPTPRM entry.
4. Only CORD1R and CORD2R can be used as a referenced coordinate system to specify topology manufacturing constraints. Only one reference coordinate system CID is allowed for each TOPVAR entry.
5. One, two or three different mirror symmetry planes can present (such as MS1=XY, MS2=YZ, and MS3=ZX).
6. Casting ("CAST") and Extrusion ("EXT") manufacturability constraints can be applied to PTYPE="PSOLID" only. Casting constraints cannot be combined with extrusion constraints for the same TOPVAR entry.
7. Some symmetry constraint types can be combined with casting or extrusion constraints. The referenced coordinate system CID must be the same for the combined constraints. Some possible combinations are:
  - For "EXT" constraints, possible combinations are (ED=X, MSi=XY, and/or ZX or CS=X), (ED=Y, MSi=YZ, and/or XY or CS=Y), (ED=Z, MSi=ZX, and/or YZ or CS=Z).
  - For "CAST" constraints, possible combinations are (DD=X or X-, MSi=XY and/or ZX or CS=X), (DD=Y or Y-, MSi=YZ and/or XY or CS=Y), (DD=Z or Z-, MSi=ZX and/or YZ or CS=Z).
8. For two dies option (DIE=2), the splitting plane is optimized. For a single die DIE=1, the parting plane is the bottom surface of the designed part in the draw direction.
9. The first symmetry segment starts at the X-axis when CS = Z (at Z-axis when CS = Y, and at the Y-axis when CS = X). One cycle symmetry can be combined with one mirror symmetry constraint as long as the axis of cyclic symmetry is normal to the plane of mirror symmetry. For example, MSi = YZ and CS = X, MSi = XZ and CS = Y, and MSi = XY and CS = Z. This feature can also be used for < 360 degrees but NCS must be given in 360 degrees.
10. It is recommended to use aligned mesh for casting property due to smaller tolerance used.
11. Without a TDMIN continuation line, the minimum member size constraint is taken from the specification of TDMIN parameter on the DOPTPRM entry. This option is applied on 2 and 3 D elements only. Minimum member size constraints can be used with "SYM", "CAST", and "EXT" constraints.

12. TVMIN and TVMAX are dimensional quantities. A guideline is that TVMIN be at least three times a representative element dimension. TVMAX must be greater than TVMIN and it is recommended that it be twice as big. If TVMAX is blank, no maximum member size is imposed. It is recommended that TVMIN always be used when TVMAX is specified.
13. “STRESS” limits can only be used for PTYPE=PSOLID and PSHELL referencing MAT1 only. The Von Mises stress (at element center) apply to all solid and/or shell elements in both designed and non-designed regions. All TOPVAR entries must have the same STLIM.
14. The TOPVAR entry cannot be used with thermal loads.
15. For normal mode topology optimization, lower and higher mode may switch during optimization. This often occurs while maximizing or constraining the first eigenfrequency. This leads to a diverging solution. A workaround is using the mean value of a few of the lowest eigenfrequency (3-6) by DRESP2.
16. The CASI solver is strongly recommended for solid elements topology problems for efficiency.
17. “PRINT” (overhang constraints) can be used for PTYPE=PSOLID only and limited to one TOPVAR entry. The result is mesh dependent. If used with minimum member size, the location of reference coordinate system may have some influence on the optimal design.

**TRIM****Trim Variable Constraint**

Specifies constraints for aeroelastic trim variables.

Format:

| 1    | 2      | 3    | 4      | 5      | 6   | 7      | 8   | 9    | 10 |
|------|--------|------|--------|--------|-----|--------|-----|------|----|
| TRIM | SID    | MACH | Q      | LABEL1 | UX1 | LABEL2 | UX2 | AEQR |    |
|      | LABEL3 | UX3  | -etc.- |        |     |        |     |      |    |

Example:

|      |      |     |      |       |     |        |     |     |  |
|------|------|-----|------|-------|-----|--------|-----|-----|--|
| TRIM | 1    | 0.9 | 100. | URDD3 | 1.0 | ANGLEA | 7.0 | 0.0 |  |
|      | ELEV | 0.2 |      |       |     |        |     |     |  |

| Descriptor | Meaning   |
|------------|---|
| SID        | Trim set identification number. (Integer > 0). See remarks 1 and 2.   |
| MACH       | Mach number. (Real $\geq 0.0$ and $\neq 1.0$ )  |
| Q          | Dynamic pressure. (Real $> 0.0$ )   |
| LABELi     | The label identifying aerodynamic trim variables defined on an AESTAT or AESURF entry. (Character)  |
| UXi        | The magnitude of the aerodynamic extra point degree-of-freedom. (Real)  |
| AEQR       | Flag to request a rigid trim analysis (Real $\geq 0.0$ and $\leq 1.0$ ; Default =1.0). A value of 0.0 provides a rigid trim analysis, see Remark 5. |

Remarks:

1. The TRIM entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all the TRIM and TRIM2 entries.
3. The selected TRIM entry specifies the constrained values of the listed extra point degrees-of-freedom (“trim variables”) for a particular loading condition. These variables are listed on AESTAT and/or AESURF entries.
4. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.
5. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

**TRIM2**

## Trim Variable Definition

Defines the state of the aerodynamic extra points for a trim analysis. All undefined extra points will be set to zero.

Format:

| 1     | 2      | 3      | 4      | 5      | 6      | 7 | 8 | 9    | 10 |
|-------|--------|--------|--------|--------|--------|---|---|------|----|
| TRIM2 | SID    | MACH   | Q      |        |        |   |   | AEQR |    |
|       | LABEL1 | VALUE1 | LABEL2 | VALUE2 | -etc.- |   |   |      |    |

Example:

|       |       |     |        |      |      |     |  |  |  |
|-------|-------|-----|--------|------|------|-----|--|--|--|
| TRIM2 | 1     | 0.9 | 100.   |      |      |     |  |  |  |
|       | URDD3 | 1.0 | ANGLEA | FREE | ELEV | 0.2 |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Trim set identification number (Integer>0). See Remarks 1. and 2.  |
| MACH       | Mach number. (Real $\geq 0.0$ and $\neq 1.0$ ). See Remark 5.  |
| Q          | Dynamic pressure. (Real $> 0.0$ )  |
| AEQR       | Flag to request a rigid trim analysis (Real $\geq 0.0$ and $\leq 1.0$ ; Default = 1.0). A value of 0.0 provides a rigid trim analysis. See Remark 6.                       |
| LABELi     | The label identifying aerodynamic trim variables defined on an AESTAT, AESURF, or AEPARM entry (Character)   |
| VALUEi     | The value assigned to LABELi. Either a real number that indicates the variable's fixed value, or one of the following words: FREE, LINKED, or SCHED. See Remarks 3. and 4. |

#### Remarks:

1. The TRIM2 entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all TRIM and TRIM2 entries.
3. A value of FREE indicates that the controller value will be solved for by the trim process. A value of LINKED indicates that the controller value will be set by an AELINK entry. A value of SCHED indicates that the controller value will be set by a CSSCHD entry. The LINKED and SCHED inputs are optional and provided as a convenience to the user. Nastran will determine the linked and scheduled controller states from the AELINK and CSSCHD entries, respectively.
4. All aerodynamic extra points that have not been defined on a TRIM2, AELINK, or CSSHED entry will be fixed to a value of zero.
5. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.

6. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

**TRMCPL**

## Trim Component Interface Coupling Parameters

Defines the interface coupling parameters for computing interface coupling matrices.

**Format:**

| 1      | 2   | 3     | 4     | 5       | 6       | 7       | 8       | 9 | 10 |
|--------|-----|-------|-------|---------|---------|---------|---------|---|----|
| TRMCPL | TID | CTYPE | PLTOL | GAPTOL1 | GAPTOL2 | GAPTOL3 | GAPTOL4 |   |    |

**Example:**

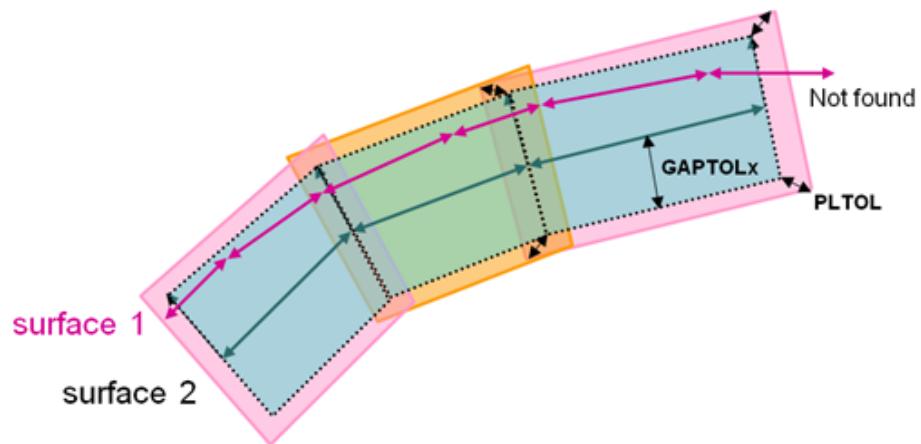
|        |   |        |      |   |  |  |  |  |  |
|--------|---|--------|------|---|--|--|--|--|--|
| TRMCPL | 1 | SGLUED | 0.12 | 5 |  |  |  |  |  |
|--------|---|--------|------|---|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| TID        | Identification number of trim component. (Integer>0; Required)   |
| CTYPE      | Interface coupling type; see Remark 2. (Character, "SGLUED", "SSLIDE", "SOPEN", "SIMPER" or "SAIRGAP"; Required)   |
| PLTOL      | Relative tolerance for in-plane/in-element/normal acceptance with respect to CTYPE. (Real>0.0; Default=0.1)  |
| GAPTOLx    | <ul style="list-style-type: none"> <li>■ GAPTOL1 Absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=0.01)</li> <li>■ GAPTOL2 Second absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL1)</li> <li>■ GAPTOL3 Third absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL2)</li> <li>■ GAPTOL4 Fourth tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL3)</li> </ul> |

**Remarks:**

1. For each interface coupling type, CTYPE, this entry is used to compute the interface matrix with incongruent meshes at the interface.
2. For the meanings of values on CTYPE, please see Bulk Data entry, ACPEMCP for the detailed descriptions.
3. TRMCPL is not a required entry for any trim component referenced by TRIMGRP and should be placed in the main bulk data section or under 'BEGIN BULK'.
4. GAPTOLx is the absolute tolerance for mapping surfaces of (structure, trim components) and (trim components, cavity). Grids on surfaces that are separated further than GAPTOLx will not be considered for coupling. GAPTOLx are utilized for successive projections. GAPTOL1 will be used for first projection. Then, GAPTOL2, GAPTOL3and GAPTOL4, if provided, will be used for subsequent projections for the remaining unprojected grids. GAPTOLx is in the same unit as the one used for length.

5. PLTOL is relative tolerance for computing volume contributions. PLTOL is in percentage.



**TSTEP****Transient Time Step**

Defines time step intervals at which a solution will be generated and output in transient analysis.

**Format:**

| 1     | 2   | 3      | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|-------|-----|--------|-----|-----|---|---|---|---|----|
| TSTEP | SID | N1     | DT1 | NO1 |   |   |   |   |    |
|       |     | N2     | DT2 | NO2 |   |   |   |   |    |
|       |     | -etc.- |     |     |   |   |   |   |    |

**Example:**

|       |   |    |      |   |  |  |  |  |  |
|-------|---|----|------|---|--|--|--|--|--|
| TSTEP | 2 | 10 | .001 | 5 |  |  |  |  |  |
|       |   | 9  | 0.01 | 1 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SID        | Set identification number. (Integer > 0)   |
| Ni         | Number of time steps of value DTi. (Integer $\geq 1$ )   |
| DTi        | Time increment. (Real > 0.0)   |
| NOi        | Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0; Default = 1) |

**Remarks:**

1. TSTEP entries must be selected with the Case Control command TSTEP = SID.
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value .001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for t = 0.0, .005, .01, .02, .03, etc.
3. See [Guidelines and Tools for Effective Dynamic Analysis](#) in *MSC Nastran Dynamic Analysis User's Guide* or a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.
5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the entry.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

**TSTEPNL****Parameters for Nonlinear Transient Analysis**

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, 600 and SOLs 400 and 700.

**Format:**

| 1       | 2      | 3      | 4     | 5      | 6      | 7     | 8       | 9       | 10 |
|---------|--------|--------|-------|--------|--------|-------|---------|---------|----|
| TSTEPNL | ID     | NDT    | DT    | NO     | METHOD | KSTEP | MAXITER | CONV    |    |
|         | EPSU   | EPSP   | EPSW  | MAXDIV | MAXQN  | MAXLS | FSTRESS |         |    |
|         | MAXBIS | ADJUST | MSTEP | RB     | MAXR   | UTOL  | RTOLB   | MINITER |    |

**Example:**

|         |       |       |       |      |       |     |     |    |  |
|---------|-------|-------|-------|------|-------|-----|-----|----|--|
| TSTEPNL | 250   | 100   | .01   | 1    | ADAPT | 2   | 10  | PW |  |
|         | 1.E-2 | 1.E-3 | 1.E-6 | 2    | 10    | 2   | .02 |    |  |
|         | 5     | 5     | 0     | 0.75 | 16.0  | 0.1 | 20. |    |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification number. (Integer > 0)  |
| NDT        | Number of time steps of value DT. See Remark 2. (Integer $\geq 3$ )   |
| DT         | Time increment. See Remark 2. (Real $> 0.0$ )   |
| NO         | Time step interval for output. Every NO-th step will be saved for output. See Remark 3. (Integer $\neq 0$ ; Default = 1)  |
| METHOD     | Method for stiffness matrix update and the transient time integration schemes. For SOLs 129 and 159, only METHOD=ADAPT is allowed and it is also the default. For SOL 400, all options are allowed. The default is AUTO with non-contact analysis and FNT with contact analysis. See Remark 4. (Character: "AUTO", "ITER", "ADAPT", "SEMI", "FNT", or "PFNT") |
| KSTEP      | KSTEP is the number of converged bisection solutions between stiffness updates for ADAPT method. (Default = 2). It is the number of iterations before stiffness updates for ITER method (Default = 10). See Remarks 18. and 19. (Integer $> 0$ ).   |
| MAXITER    | Limit on number of iterations for each time step. See Remarks 5., 6., and 18. (Integer $\neq 0$ ; Default = 10 for ADAPT method and 25 for non-ADAPT methods)   |
| CONV       | Flags to select convergence criteria. See Remarks 7. and 21. (Character; Default = "PW" for SOLs 129 and 159, "UPW" for SOL 400 with non-contact analysis, and "PV" for SOL 400 with contact analysis.)   |
| EPSU       | Error tolerance for displacement (U) criterion. See Remark 20. (Real $\neq 0.0$ ; Default = 1.0E-2 for all methods except PFTN. For PFTN, Default = -1.0E-2)  |
| EPSP       | Error tolerance for load (P) criterion. (Real $> 0.0$ ; Default = 1.0E-3 for SOLs 129 and 159, 1.0E-2 for SOL 400)  |

| Descriptor | Meaning  |
|------------|--|
| EPSW       | Error tolerance for work (W) or energy criterion. See Remark 20. (Real > 0.0; Default = 1.0E-6 for SOLs 129 and 159, 1.0E-2 for SOL 400 and non-PFNT methods, -1.0E-2 for SOL 400 and PFNT method)                     |
| MAXDIV     | Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark 8. (Integer ≠ 0; Default = 2)  |
| MAXQN      | Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 9. (Integer ≥ 0; Default = 10 for all methods except PFNT. For PFNT, Default = 0)  |
| MAXLS      | Maximum number of line searches allowed per iteration. See Remark 9. (Integer ≥ 0; Default = 2 for all methods except PFNT. For PFNT, Default = 0)   |
| FSTRESS    | Fraction of effective stress ( $\bar{\sigma}$ ) used to limit the subincrement size in the material routines. See Remark 10. (0.0 < Real < 1.0; Default = 0.2)   |
| MAXBIS     | Maximum number of bisections allowed for each time step. See Remark 11. and 12. (-9 ≤ Integer ≤ 9; Default = 5 except for MAXITER < 0 and SOL 400. For MAXITER < 0 and SOL 400, Default = 0)                           |
| ADJUST     | Time step skip factor for automatic time step adjustment. See Remark 13. (Integer ≥ 0; Default = 5)  |
| MSTEP      | Number of steps to obtain the dominant period response. See Remark 14. (10 ≤ Integer ≤ 200; Default = variable between 20 and 40 for SOL 129 and between 10 and 20 for SOL 400.)                                       |
| RB         | Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 14. (0.1 ≤ Real ≤ 1.0; Default = 0.6)   |
| MAXR       | Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 15. (Real ≥ 1.0; Default = 32.0)   |
| UTOL       | Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 16. (0.001 < Real ≤ 1.0; Default = 0.1)   |
| RTOLB      | Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 17. (Real > 2.0; Default = 20.0)  |
| MINITER    | Minimum number of iterations for a load increment, SOL 400 only. (Default = 1 except for the contact analysis. For contact analysis, Default = 2) It is recommended to set MINITER = 2 when high accuracy is expected) |

**Remarks:**

- For SOL 129 and SOL 159, the Bulk Data entry is selected by the Case Control command = ID. Each residual structure subcase requires a entry and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase defined by the end conditions of the previous subcase.

For SOL 400, the is selected in the STEP ID's listed within a SUBCASE ID. For SOL 400 each SUBCASE is independent of the previous statement.

2. NDT is used to define the total duration for analysis, which is NDT \* DT. Since DT is adjusted during the analysis, the actual number of time steps, in general, will not be equal to NDT). Also, DT is used only as an initial value for the time increment.
3. For printing and plotting the solution with SOLs 129 and 159, data recovery is performed at time 0, NO\*DT, 2\*NO\*DT, ..., and the last converged step, where DT is internally computed time increment and may change at every time step. For SOL 400 and NO > 0, data recovery is performed at time 0, NO\*DTI, 2\*NO\*DTI, ..., and the last converged step, where DTI is the user input initial time increment and it is a constant. For SOL 400 thermal analysis and NO < 0, the SOL 159 scheme is used. For SOL 400 structural analysis and NO < 0, the SOL 129 scheme is used for SOL 400, i.e., data recovery is performed at time 0, |NO|\*DT, 2\*|NO|\*DT, ..., and the converged step. See Remark 13. on how DT is changed.
4. The stiffness update strategy as well as the transient integration method is selected in the METHOD field.
  - a. METHOD=“ADAPT”: The program automatically adjusts the incremental time and uses bisection. During the bisection process, the stiffness matrix is updated every KSTEPth converged bisection solution. This is the **only** method available for SOLs 129 and 159 and is also their **default**.
  - b. METHOD=“AUTO”: The stiffness matrix is automatically updated to improve the convergence. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0. KSTEP value is ignored during iteration. This is the **default** method for SOL 400.
  - c. METHOD = “ITER”: The stiffness is updated at every KSTEPth iterations. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0.
  - d. METHOD=“SEMI”: Same as the AUTO method except that the stiffness updated at the first iteration, and then starts the AUTO iteration scheme.
  - e. METHOD = “FNT”: This is the Full Newton iteration method; the stiffness is updated at every iteration. In comparison with the PFNT method, the defaults for FNT are EPSU = 0.01, EPSW = 0.01 and MAXLS = 2. See Remark 19.
  - f. METHOD = “PFNT”: This is the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are EPSU = -0.01, EPSW = -0.01, and MAXLS = 0. See Remark 19.
5. This remark applies to SOLs 129 or 159 only. The number of iterations for a time step is limited to MAXITER. If MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process. See Remark 8.

6. This remark applies to SOL 400 only. The number of iterations for a load increment is limited to  $|MAXITER|$ . If the solution does not converge in  $|MAXITER|$  iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or  $MAXBIS = 0$ ) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution still diverges. If MAXDIV is negative, the analysis is terminated immediately.
- If MAXITER is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each time step. The default for  $MAXBIS = 1$ , if  $MAXITER < 0$ . Also for SOL 400, the value of MAXITER for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.
7. The convergence test flags (U = displacement error test, P = load equilibrium error test, W = work error test, V = vector component method, N = Length method) and the error tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All requested criteria (combination of U, P, W, V, and/or N) are satisfied upon convergence. Note that at least two iterations are necessary to check the displacement convergence criterion. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For V and N, see Remark 21.
8. MAXDIV provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions (NDIV) is incremented by 1 or 2. The solution is assumed to diverge when NDIV reaches MAXDIV during the iteration. If the bisection option is used (allowed MAXBIS times) the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new stiffness based on the converged state at the beginning of the time step. If NDIV reaches MAXDIV again within the same time step, the analysis is terminated for SOLs 129 and 159. For SOL 400, the termination of analysis is dependent on the sign of MAXDIV. If MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution is still diverges. If MAXDIV is negative, the analysis is terminated immediately. See Remark 6.
9. Nonzero values of MAXQN and MAXLS will activate the quasi-Newton update and the line search process, respectively.
10. The number of subincrements in the material routines is determined such that the subincrement size is approximately  $FSTRESS \cdot \bar{\sigma}$ . FSTRESS is also used to establish a tolerance for error correction in elastoplastic material, i.e., error in yield function  $< FSTRESS \cdot \text{yield stress}$
- If the limit is exceeded at the converging state, the program will terminate with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface, resulting in  $\delta = 0$ .
11. The remark applies to SOLs 129 or 159 only. The bisection process is activated when divergence occurs and  $MAXBIS \neq 0$ . The number of bisections for a time increment is limited to  $|MAXBIS|$ . If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in  $|MAXBIS|$  bisection, the analysis is terminated.

12. This remark applies to SOL 400 only. The bisection process is activated when divergence occurs and  $\text{MAXBIS} \neq 0$ . The number of bisections for a time step is limited to the absolute value of  $\text{MAXBIS}$ . Different actions are taken when the solution diverges depending on the sign of  $\text{MAXBIS}$ . If  $\text{MAXBIS}$  is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If  $\text{MAXBIS}$  is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after  $|\text{MAXBIS}|$  bisections, the analysis is continued or terminated depended on the sign of  $\text{MAXDIV}$ . See Remark 8.
13. ADJUST controls the automatic time stepping. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is the user's responsibility to ensure that the loading history is properly traced with the ADJUST option. The ADJUST option should be suppressed for the duration of short pulse loading. In particular, for impact problems in SOL 400, it is recommended that the ADJUST option be suppressed since the short duration pulses during the impact may not be tracked well by the frequency-based algorithm. It should also be noted that the TZEROMAX process (where the step is repeated with the same/smaller time step after the first 2 increments) is not currently available for contact problems. If unsure, start with a value for DT that is much smaller than the pulse duration in order to properly represent the loading pattern.
  - If  $\text{ADJUST} = 0$ , then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
  - If  $\text{ADJUST} > 0$ , the time increment is continually adjusted for the first few steps until a good value of  $\Delta t$  is obtained. After this initial adjustment, the time increment is adjusted every  $\text{ADJUST}$ -th time step only.
  - For SOL 400, if  $\text{ADJUST} > 0$  and  $\text{NO} > 0$  (see Remark 3.), the analysis time step can reduce without limit, but it cannot increase more than  $\text{DT} * \text{NO}$ . This means that, if  $\text{NO} = 1$ , the analysis time step can only reduce, but cannot increase. The user should use  $\text{NO} > 1$  to take the advantage of the range of automatic time stepping. If  $\text{NO} < 0$ , the time step can both increase and decrease without limit. But in this case, the user will not know the exact output locations.
  - If  $\text{ADJUST}$  is one order greater than  $\text{NDT}$ , then automatic adjustment is deactivated after the initial adjustment.
14. MSTEP and RB are used to adjust the time increment during analysis. The recommended value of MSTEP is 10 to 40. A larger value (e.g., 40) is required for highly nonlinear problems. By default, the program automatically computes the value of MSTEP based on the changes in the stiffness.

The time increment adjustment is based on the number of time steps desired to capture the dominant frequency response accurately. The time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r)\Delta t_n$$

where

$$r = \frac{1}{\text{MSTEP}} \left( \frac{2\pi}{\omega_n} \right) \left( \frac{1}{\Delta t_n} \right)$$

where:

|            |                                |
|------------|--------------------------------|
| $f = 0.25$ | for $r < 0.5 \cdot RB$         |
| $f = 0.5$  | for $0.5 \cdot RB \leq r < RB$ |
| $f = 1.0$  | for $RB \leq r < 2.0$          |
| $f = 2.0$  | for $2.0 \leq r < 3.0 / RB$    |
| $f = 4.0$  | for $r \geq 3.0 / RB$          |

15. MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\text{MIN}\left(\frac{DT}{2^{\text{MAXBIS}}}, \frac{DT}{\text{MAXR}}\right) \leq \Delta t \leq \text{MIN}(\text{MAXR} \cdot DT, NO \cdot DT)$$

16. UTOL is a tolerance used to filter undesirable time step adjustments; i.e.,

$$\frac{\|\dot{U}_n\|}{\|\dot{U}\|_{max}} < \text{UTOL}$$

Under this condition no time step adjustment is performed in a structural analysis (SOL 129). In a heat transfer analysis (SOL 159) the time step is doubled.

17. The bisection is activated if the incremental rotation for any degree-of-freedom ( $\Delta\theta_x, \Delta\theta_y, \Delta\theta_z$ ) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.
18. For non-ADAPT (except FNT and PFNT) methods, the stiffness will be updated at convergence if the number of iterations since last iteration is equal or greater than KSTEP. In addition, for SOL 400 and ADAPT method, the stiffness will be updated if  $3^*\text{MAXITER}$  iterations are performed. For SOL 400 and non-ADAPT methods, the stiffness will be updated if MAXITER iterations are performed.
19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed file with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.
21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always

negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV and CONV=D is the same as CONV=UPD. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

22. For SOL 700, only fields ID, NDT, and DTare used.
23. In NLTRAN analysis, the output is also influenced by NLPACK. Please refer to NLPACK for an example.

**TTEMP**

## Temperature Distribution of Transient Response

Defines a time-dependent temperature distribution for the nonlinear transient analysis in SOL 400.

$$\{T(t)\} = \{A(T(x)) \cdot F(t)\}$$

where  $A(T(x))$  defines a spatial temperature distribution and  $F(t)$  a time function.  $T(t)$  is the time dependent temperature distribution for use in the nonlinear elements in nonlinear transient analysis.

**Format:**

| 1     | 2   | 3        | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----------|-----|---|---|---|---|---|----|
| TTEMP | SID | GROUP_ID | TID |   |   |   |   |   |    |

**Example:**

|       |    |     |    |  |  |  |  |  |  |
|-------|----|-----|----|--|--|--|--|--|--|
| TTEMP | 11 | 101 | 31 |  |  |  |  |  |  |
|-------|----|-----|----|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SID        | Temperature set identification number. (Integer > 0)                     |
| GROUP_ID   | Temperature group identification number (Integer > 0 or ==1)             |
| TID        | Identification number of TABLEDi entry that gives $F(t)$ . (Integer > 0) |

**Remarks:**

1. The temperature distribution TTEMP must be selected by a Case Control command TEMP(LOAD)=SID in order to be used in the nonlinear transient analysis.
2. This command is used in SOL 400 only when ANALYSIS=NLTTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0). There should be only one temperature set for each STEP.
3. GROUP\_ID determines the distribution of temperatures. It references the TMPSET Bulk Data entry to define all grid points, which reference the same TABLEDi entry. Each grid point can have its own GROUP\_ID if necessary. GROUP\_ID==1 means all grid points are in one group and reference the same TTEMP Bulk Data entry.
4. If the TEMP(INIT) Case Control command references a TTEMP entry, then only the spatial distribution of the TTEMP will be used as the initial temperature distribution for the TEMP(INIT) command.
5. TTEMP may be used in nonlinear transient analysis in SOL 400. For pure linear analysis in SOL 400, DLOAD/TLOADi has to be used to define a time-dependent temperature distribution.

**UDNAME****User Defined File Name**

Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD, TABLRPC, and MATDIGI.

**Format:**

| 1      | 2    | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|------|---|---|---|---|---|---|---|----|
| UDNAME | UDID |   |   |   |   |   |   |   |    |
|        | NAME |   |   |   |   |   |   |   |    |

**Example:**

|        |            |  |  |  |  |  |  |  |  |
|--------|------------|--|--|--|--|--|--|--|--|
| UDNAME | 10         |  |  |  |  |  |  |  |  |
|        | myfile.dat |  |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| UDID       | Unique UDID (Integer>0). See Remark 1.   |
| NAME       | Name of a file (with or without path) such as the external loading time history in DAC, RPC, or S3T format or DIGIMAT material file (Character). |

**Remarks:**

1. The UDID is referenced by FTGLOAD, TABLRPC, or MATDIGI entries
2. The NAME is limited to a maximum of 256 characters ( corresponding to 4 lines of data in fields 2-9 ). For Fatigue analysis using .rsp (RPC) files, there is a 128 character limit (corresponding to 2 lines of data in fields 2-9). Embedded blanks are not supported.
3. If only a NAME with no path (e.g., *sine01.dac*) is supplied, the file is assumed to be located in the same directory as the Nastran input file. If an absolute or relative path is supplied (e.g., */local/user/fatigue/sine01.dac*), it will be used.

**UDSESV**

Define the number and names of user state variables for material user subroutines  
(SOL 400 only)

Define the number and names of the user state variables. Each state variable has default nominal name if a user name is not given. This is a global entry.

Format:

| 1      | 2   | 3        | 4    | 5        | 6   | 7        | 8   | 9        | 10 |
|--------|-----|----------|------|----------|-----|----------|-----|----------|----|
| UDSESV |     | NSTATS   |      |          |     |          |     |          |    |
|        | SV2 | SV2_NAME | SV3  | SV3_NAME | SV4 | SV4_NAME | SV5 | SV5_NAME |    |
|        | SV6 | SV6_NAME | .ETC |          |     |          |     |          |    |

Example:

|        |     |      |     |      |  |  |  |  |  |
|--------|-----|------|-----|------|--|--|--|--|--|
| UDSESV |     | 3    |     |      |  |  |  |  |  |
|        | SV2 | VAR2 | SV3 | VAR3 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| NSTATS     | The number of user defined state variables. (Integer $\geq 1$ )  |
| SVi        | The default nominal name of state variable (CHARACTER, $i \geq 1$ , where $i$ is the index number of the state variable) |
| SVi_NAME   | The state variable name defined by user (CHARACTER, Default = SVi, where $i$ is the index number of the state variable)  |

Remarks:

1. This is a global entry that defines user state variables for material user subroutines. The temperature will always be passed to material use subroutine as the first state variable; its name should not be redefined in this entry.
2. If a state variable is not given a name, SVi will be used as its name. The number i is the index number of the state variable.
3. For output, either state variables names given in UDSESV or default SVi names can be used in NLOUT entry in case control. The state variables names will be used as keywords for output selection.
4. The 1st state variable is always temperature. The remaining user defined state variables are defined and used only by user, Nastran will not use them.
5. This entry only supports UMAT and UCOHES user subroutines, see UNAME in MATUDS
6. This entry is required when MATDIGI is defined to define the number of state variables.
7. The number of state variables is limited to 100 state variables (including temperature) for SOL400.

3296

**UNBALNC**

Specifies an Unbalanced Load for Rotordynamic Transient or Frequency Response Analysis

**UNBALNC**

Specifies an Unbalanced Load for Rotordynamic Transient or Frequency Response Analysis

Used in rotordynamic analyses to specify a rotating unbalance load in terms of a cylindrical system with the rotor rotation as the z-axis.

**Format:**

| 1       | 2       | 3     | 4       | 5                 | 6                  | 7     | 8 | 9 | 10 |
|---------|---------|-------|---------|-------------------|--------------------|-------|---|---|----|
| UNBALNC | SID     | MASS  | GRID    | X1                | X2                 | X3    |   |   |    |
|         | ROFFSET | THETA | ZOFFSET | T/F <sub>on</sub> | T/F <sub>OFF</sub> | CFLAG |   |   |    |
|         | UFT1    | UFT2  | UFT3    | UFR1              | UFR2               | UFR3  |   |   |    |
|         | MCT1    | MCT2  | MCT3    | MCR1              | MCR2               | MCR3  |   |   |    |
|         | SCR1    | SCR2  | SCR3    |                   |                    |       |   |   |    |

**Example:**

|         |      |      |      |      |      |      |  |  |  |
|---------|------|------|------|------|------|------|--|--|--|
| UNBALNC | 100  | 0.1  | 1001 | 0.0  | 1.0  | 0.0  |  |  |  |
|         | 0.02 | 30.0 | 0.5  |      |      | MASS |  |  |  |
|         | 1001 | 1002 | 1003 | 1004 | 1005 | 1006 |  |  |  |
|         | 2001 | 2002 | 2003 | 2004 | 2005 | 2006 |  |  |  |
|         | 3001 | 3003 | 3004 |      |      |      |  |  |  |

| Descriptor         | Meaning  |
|--------------------|--|
| SID                | Set identification number. (Integer; Required; no Default)   |
| MASS               | Mass of imbalance. See Remark 4. (Real or Integer; if integer must be > 0; Required; no Default)   |
| GRID               | Grid identification number of applying imbalance. (Integer; Required; no Default)  |
| X1, X2, X3         | Components of the vector, from GRID, in the displacement coordinate system of GRID, which is used to define a cylindrical coordinate system centered at GRID; see Remark 5. (Real; Required; no Default) |
| ROFFSET            | Offset of mass in the radial direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 1.0)   |
| THETA              | Angular position, in degrees, of the mass in the unbalance coordinate system. (Real; Default = 0.0)  |
| ZOFFSET            | Offset of mass in the z-direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 0.0)  |
| T/F <sub>ON</sub>  | Start time or frequency for applying imbalance load. See Remark 6. (Real $\geq 0.0$ ; Default = 0.0)   |
| T/F <sub>OFF</sub> | Stop time or frequency for terminating imbalance load. See Remark 6. (Real $> 0.0$ ; Default = 999999.0)   |

| Descriptor | Meaning  |
|------------|--|
| UFT1-3     | EPOINTS to output the unbalanced forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)  |
| UFR1-3     | EPOINTS to output the unbalanced forces in the R1, R2, and R3 directions. See Remark 6. (Integer > 0)  |
| MCT1-3     | EPOINTS to output the mass-correction forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)   |
| MCR1-3     | EPOINTS to output the mass-correction forces in the R1, T2, and R3 directions. See Remark 6. (Integer > 0)   |
| SCR1-3     | EPOINTS to output the speed-correction forces in the R1, R2, and R3 directions. (Integer > 0)  |
| CFLAG      | Correct flag to specify whether 1) the mass will be used to modify the total mass in the transient response calculations, 2) the effect of the rotor spin rate change will be included in the transient response calculation, or 3) both; see Remark 7. (Character: NONE, MASS, SPEED, or BOTH, Default = NONE). |

#### Remarks:

1. Multiple UNBALNC entries with the same SID value are allowed.
2. For transient response, UNBALNC is specified by the RGYRO Case Control command. For frequency response, it is specified by the DLOAD Case Control command.
3. The imbalance load will be generated based on the mass value, offset values, and the rotor spin speed.
4. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a TABLEDi entry that specifies the value as a function of time for transient response or frequency for frequency response.
5. A cylindrical coordinate system is used to determine the initial position and rotation direction of the mass unbalance. Theta is measured from the plane defined by the z-axis and the user specified vector (X1, X2, X3). Theta = 0.0 is in the direction of the user-specified vector (X1, X2, X3). Unbalance rotation is in the positive z-direction.
6. For frequency response analysis, the EPOINTS and the continuation cards are ignored.
7. If the mass loss (increase) is relatively small, the correction may safely be ignored. (CFLAG = NONE). These error corrections terms are applied for analysis in fixed reference frame only.
8. The mass specified on UNBALNC is scaled with PARAM, WTMASS factor for both transient and frequency response analysis.

Equations in fixed reference frame:

- a. For Frequency response analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} mr\omega^2(\cos\theta_k + i\sin\theta_k) \\ mr\omega^2(\sin\theta_k + i\cos\theta_k) \end{Bmatrix}$$

b. For transient analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} m(t)r(t)(\Omega_j^2(t)\cos\theta_{jk}(t) + \dot{\Omega}_j(t)\sin\theta_{jk}(t)) \\ m(t)r(t)(\Omega_j^2(t)\sin\theta_{jk}(t) - \dot{\Omega}_j(t)\cos\theta_{jk}(t)) \end{Bmatrix}$$

where,

- $m(t)$  Specified by the user (MASS field in UNBALNC)
- $r(t)$  Specified by the user (ROFFSET field in UNBALNC)
- $\Omega_j(t)$  Specified by the user through RSINT/RSPINR entry
- $\dot{\Omega}_j(t)$  Derived from  $\Omega_j(t)$
- $\theta_{jk}(t)$  Instantaneous angular location, measured from axis 1 to axis 2 ( $= \int_0^t \omega_j(t)dt + \theta_k$ )
- $\theta_k$  Specified by the user (THETA field in UNBALNC)
- $j$  Rotor on which the grid is located

**UNGLUE**

Defines Grids that Should Be Eliminated From Glued Contact

This entry may be used in SOLs 101 or 400 but is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact.

**Format:**

| 1      | 2   | 3    | 4   | 5    | 6   | 7      | 8 | 9 | 10 |
|--------|-----|------|-----|------|-----|--------|---|---|----|
| UNGLUE | ID  | BID  | ID1 | THRU | ID2 | BY     | N |   |    |
|        | ID3 | THRU | ID4 | ID5  | ID6 | -etc.- |   |   |    |

**Example:**

|        |    |      |      |      |      |      |      |  |  |
|--------|----|------|------|------|------|------|------|--|--|
| UNGLUE | 10 | 101  | 20   | THRU | 300  | BY   | 2    |  |  |
|        |    | 3457 | 8456 | 4712 | 1000 | THRU | 2000 |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Identification number referenced by a SUBCASE or STEP Case Control BCONTACT or UNGLUE command. See Remark 1. (Integer; no Default) |
| BID        | Identification of the specified BCBODY. (Integer > 0; no Default).   |
| IDi        | ID list of grids. (Integer > 0; no Default).   |
| THRU       | Enter THRU if a range of grids is required. (Character)  |
| BY         | Enter BY if the range of grids is not consecutive. (Character)   |
| N          | BY increment. (Integer > 0).   |

**Remarks:**

1. To place an entry in loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or UNGLUE=0 and is always automatically executed. To place an entry in any physical loadcase (SUBCASE or STEP), ID must be selected by the Case Control command BCONTACT=ID or UNGLUE=ID. Note that UNGLUE Case Control will take precedence over the BCONTACT Case Control for this entry with same ID. UNGLUE is ignored by Permanent contact.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.

## UNGLUE      Defines Grids that Should Be Eliminated From Glued Contact in SOL 600

This entry is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact. SOL 600 only.

**Format:**

| 1      | 2    | 3    | 4  | 5    | 6  | 7  | 8  | 9 | 10 |
|--------|------|------|----|------|----|----|----|---|----|
| UNGLUE | IDBC | IBOD | D1 | THRU | D2 | BY | N3 |   |    |
|        |      |      | D4 | THRU | D5 | BY | N6 |   |    |

**Example:**

|        |   |   |   |      |     |  |  |  |  |
|--------|---|---|---|------|-----|--|--|--|--|
| UNGLUE | 1 | 1 | 1 | THRU | 100 |  |  |  |  |
|--------|---|---|---|------|-----|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| IDBC       | Identification number of a Case Control BC CONTACT command defining the subcase to which these items should be available.<br>Enter 0 if this entry should apply to Marc's increment zero. Enter -9999 if it should apply to all subcases. (Integer; no Default)                            |
| IBOD       | Identification number of a matching BCBODY Bulk Data entry of a flexible surface defining the body with GRIDS to be removed from glued contact. The BCBODY may include additional grids (not included in this entry) that remain glued. (Integer; Default = 1) See Remarks 4. and 5. (3,1) |
| IDi        | ID list of grids. (Integer > 0; no Default).   |
| THRU       | Enter THRU if a range of grids is required. (Character)  |
| BY         | Enter BY if the range of grids is not consecutive. (Character)   |
| N          | BY increment. (Integer > 0).   |

**Remarks:**

1. Repeat the continuation entry until all grids for the particular body are described.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.
3. This entry corresponds to Marc's DEACT GLUE option. Items (i,j) indicate the corresponding data block and field.
4. In certain models, there are no BCBODY entries (for example; self contact) because the entire model comprises one body. For such models IBOD may be left blank.

5. If IBOD is a positive number, it will be converted to the body number using the BCBODY entries.  
To override this conversion and use the body number directly, enter IBOD as a negative number whose magnitude is equal to the body number desired.
6. Multiple UNGLUE entries with the same IDBC are not allowed.

**USET****Degree-of-Freedom Set Definition**

Defines a degree-of-freedom set.

**Format:**

| 1    | 2     | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|------|-------|-----|----|-----|----|-----|----|---|----|
| USET | SNAME | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|      |    |     |    |    |   |  |  |  |  |
|------|----|-----|----|----|---|--|--|--|--|
| USET | U4 | 333 | 26 | 17 | 0 |  |  |  |  |
|------|----|-----|----|----|---|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| SNAME      | Set name. (One to four characters, or the string “ZERO” followed by the set name.)   |
| IDi        | Grid or scalar point identification number. (Integer > 0)  |
| Ci         | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |

**Remarks:**

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. A maximum of 18 degrees-of-freedom may be designated on a single entry.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.
6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

**USET1**

## Degree-of-Freedom Set Definition, Alternate Form

Defines a degrees-of-freedom set.

**Format:**

| 1     | 2     | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-------|-----|--------|-----|-----|-----|-----|-----|----|
| USET1 | SNAME | C   | ID1    | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|       | ID7   | ID8 | -etc.- |     |     |     |     |     |    |

**Example:**

|       |    |     |   |   |    |   |   |   |  |
|-------|----|-----|---|---|----|---|---|---|--|
| USET1 | SB | 345 | 2 | 1 | 36 | 5 | 9 | 7 |  |
|       | 40 |     |   |   |    |   |   |   |  |

**Alternate Format and Example:**

|       |       |     |     |        |     |  |  |  |  |
|-------|-------|-----|-----|--------|-----|--|--|--|--|
| USET1 | SNAME | C   | ID1 | “THRU” | ID2 |  |  |  |  |
| USET1 | SB    | 123 | 170 | THRU   | 180 |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| SNAME      | Set name. (One to four characters or the word “ZERO” followed by the set name.)  |
| C          | Component numbers. (Integer zero or blank for scalar points or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDi        | Grid or scalar point identification number. (Integer > 0; for “THRU” option, ID1 < ID2.)   |

**Remarks:**

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME=“ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. If the alternate format is used, all of the points ID1 through ID2 are assigned to the set.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.

6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

**USRSUB6**

Defines User Subroutines for SOL 600

Defines user subroutines used in SOL 600 only.

**Format:**

| 1       | 2  | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|-----|----|----|----|----|----|----|----|
| USRSUB6 | U1 | U2  | U3 | U4 | U5 | U6 | U7 | U8 |    |
|         | U9 | U10 |    |    |    |    |    |    |    |

**Examples:**

|          |               |       |        |  |  |  |  |  |  |
|----------|---------------|-------|--------|--|--|--|--|--|--|
| USRSUB6  | UDAMAG        | uvoid | TENSOF |  |  |  |  |  |  |
| USRSUB6* | SEPFORBB<br>C |       |        |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| Ui         | Name of user subroutine(s) to be included. See Marc Volume D for list of available user subroutines. Do not include the .f extension on this entry, however, the actual file on the disk must have the .f extension. If any user subroutine exceeds 8 characters, use the wide field format for the primary line and all continuation lines. (Character; no Default) |

**Notes:**

1. All user subroutines must reside in the directory where the Nastran input file resides.
2. All user subroutines on disk must be in lower case and have an extension of .f. The names entered in the bulk data entry may be in upper or lower case. They will be converted to lower case.
3. SOL 600 combines all user subroutines into one large subroutine named u600.f and u600.f is passed to the Marc command line when spawned from Nastran.
4. If only one user subroutine is required, an alternate is to use PARAM,MARCUSUB,name.

**UXVEC****Control Parameter State**

Specification of a vector of aerodynamic control point (extra point) values. These data define the control positions corresponding to user defined nonlinear control forces that have been defined by AEDW, AEPRESS and AEFORCE entries. Only nonzero values need to be defined.

**Format:**

| 1     | 2      | 3   | 4      | 5   | 6      | 7 | 8 | 9 | 10 |
|-------|--------|-----|--------|-----|--------|---|---|---|----|
| UXVEC | ID     |     |        |     |        |   |   |   |    |
|       | LABEL1 | UX1 | LABEL2 | UX2 | -etc.- |   |   |   |    |

**Example:**

|       |        |      |        |      |  |  |  |  |  |
|-------|--------|------|--------|------|--|--|--|--|--|
| UXVEC | 1001   |      |        |      |  |  |  |  |  |
|       | THRUST | 1.E4 | ANGLEA | .015 |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| ID         | Control vector identification number, see Remark 1. (Integer > 0)  |
| LABELi     | Controller name. This must correspond to an existing AESURF, AESTAT or AEPARM label or INTERCPT (Character). See Remark 4. |
| UXi        | The magnitude of the aerodynamic extra point degree-of-freedom (Real)  |

**Remarks:**

1. The ID is referenced by the AEUXREF Case Control command and/or by AEDW, AEPRESS, and/or AEFORCE entries.
2. The units of the user defined AEPARM controllers are implied by their use on this entry and the corresponding values on the force vector definition. The user must be self-consistent in all uses of the same controller. AESURF controllers are expressed in radians as are the rigid body angles ANGLEA and BETA. The rigid body rates, ROLL, PITCH and YAW are nondimensional rates  $pb/2V$ ,  $qc/2V$ ,  $rb/2V$ ; respectively. V is the velocity and b and c are the reference span and chord lengths, respectively.
3. LABELs that are part of the UX vector that are omitted in the UXVEC specification are assigned a value of 0.0.
4. INTERCPT = 1.0 is implied on all UXVEC input.  
INTERCPT = 0.0 indicates that associated load is a perturbation.

## VCCT

## Virtual Crack Closure Technique - SOL 400

Format:

| 1    | 2      | 3     | 4     | 5     | 6    | 7      | 8        | 9         | 10 |
|------|--------|-------|-------|-------|------|--------|----------|-----------|----|
| VCCT | ID     | IDCR  | ITYPE | IGROW | INCM | METHOD |          |           |    |
|      | CGI    | GC    |       |       |      |        | GC-II    | GC-III    |    |
|      | TABCGI | TABGC |       |       |      |        | TABGC-II | TABGC-III |    |
|      | G1     | G2    | G3    | G4    | G5   | etc.   |          |           |    |

Example:

|      |     |       |   |   |   |   |  |   |  |
|------|-----|-------|---|---|---|---|--|---|--|
| VCCT | 101 | 1     | 2 | 2 | 2 | 1 |  | 1 |  |
|      |     | 2000. |   |   |   |   |  |   |  |
|      |     | 0     |   |   |   |   |  |   |  |
|      | 51  | 52    |   |   |   |   |  |   |  |

| Descriptor | Meaning   |
|------------|---|
| ID         | Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)   |
| IDCR       | Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR. (Integer > 0; Default = 1)  |
| ITYPE      | Type of crack propagation. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0 No crack propagation</li> <li>2 Direct crack propagation</li> </ul>  |
| IGROW      | Specifies how the crack grows. (Integer; Default = 2) <ul style="list-style-type: none"> <li>1 Uses remeshing (not presently available)</li> <li>2 Release user tyings or glued contact</li> </ul>  |
| INCM       | Specifies the crack growth increment (Integer; Default = 1) <ul style="list-style-type: none"> <li>1 Uses a fixed increment or a user subroutine</li> </ul>   |
| METHOD     | Specifies the method used for the estimated crack growth direction (Integer; Default=1) <ul style="list-style-type: none"> <li>1 Maximum hoop stress criterion (Default)</li> </ul>   |
| CGI        | Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law. (Not presently used)  |
| GC         | Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9. For SOL 600, if it is desired that the mode I crack resistance be zero, enter GC as a negative value. |

| Descriptor            | Meaning   |
|-----------------------|---|
| GC-II                 | Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.  |
| GC-III                | Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.   |
| TABC <sub>GI</sub>    | TABLEM <sub>i</sub> or TABL3D <sub>i</sub> for CGI (crack growth increment). (Integer; Default = 0)   |
| TABG <sub>C</sub>     | TABLEM <sub>i</sub> or TABL3D <sub>i</sub> for scaling GC (crack growth resistance). (Integer; Default = 0)   |
| TABG <sub>C-II</sub>  | TABLEM <sub>i</sub> or TABL3D <sub>i</sub> for scaling GC-II (Integer; Default = 0)   |
| TABG <sub>C-III</sub> | TABLEM <sub>i</sub> or TABL3D <sub>i</sub> for scaling GC-III (Integer; Default = 0)  |
| G <sub>i</sub>        | Grids along the crack front – for a shell there is only one node. See Remark 6. (Integer or “THRU” or “BY”; no Default; at least one value, G <sub>1</sub> , must be provided.) |

#### Remarks:

1. The grids entered on this entry MUST be associated with SOL400 elements that have had their capabilities extended by use of either a PSNLN1, PSNLN2, PSLDN1, PLCOMP or PCOMPLS or a combination of these entries.
2. ID corresponds to a Case Control VCCT entry.
3. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
4. This entry may be repeated as many times as necessary to describe all the cracks in the model.
5. The 4<sup>th</sup> line may be repeated as many times as necessary to describe all grids on the crack front
6. If G<sub>1</sub> is negative, the absolute value of G<sub>1</sub> is used as the ID of a SET3 entry providing the list of grids. For this case, G<sub>2</sub>, G<sub>3</sub>, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
7. The values of the entries on the second line that are not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
8. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

**VCCT****Virtual Crack Closure Technique in SOL 600**

Provides data to simulate crack growth using virtual crack closure technology methods in SOL 600.

**Format:**

| 1    | 2      | 3     | 4      | 5     | 6    | 7       | 8        | 9         | 10 |
|------|--------|-------|--------|-------|------|---------|----------|-----------|----|
| VCCT | ID     | IDCR  | ITYPE  | IGROW | INCM | METHOD  | TIME     | IACT      |    |
|      | CGI    | GC    | GTH    | C     | M    | GMIN    | GC-II    | GC-III    |    |
|      | TABCGI | TABGC | TABGTH | TABC  | TABM | TABGMIN | TABGC-II | TABGC-III |    |
|      | G1     | G2    | G3     | G4    | G5   | etc.    |          |           |    |

**Alternate Format:**

|      |        |       |        |       |      |         |          |           |  |
|------|--------|-------|--------|-------|------|---------|----------|-----------|--|
| VCCT | ID     | IDCR  | ITYPE  | IGROW | INCM | METHOD  | TIME     | IACT      |  |
|      | CGI    | GC    | GTH    | C     | M    | GMIN    | GC-II    | GC-III    |  |
|      | TABCGI | TABGC | TABGTH | TABC  | TABM | TABGMIN | TABGC-II | TABGC-III |  |
|      | G1     | THRU  | G2     | BY    | G3   |         |          |           |  |

**Example:**

|      |     |       |     |    |     |   |  |   |  |
|------|-----|-------|-----|----|-----|---|--|---|--|
| VCCT | 101 | 1     | 2   | 2  | 2   | 1 |  | 1 |  |
|      |     | 2000. | 12. | 4. | 2.0 |   |  |   |  |
|      |     | 0     | 0   | 0  | 0   |   |  |   |  |
|      | 51  | 52    |     |    |     |   |  |   |  |

| Descriptor     | Meaning  |
|----------------|--|
| ID             | Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)  |
| IDCR           | Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR and a different ID using the IACT field. (Integer > 0; Default = 1) |
| ITYPE<br>(6-1) | Type of crack propagation. (Integer; Default = 2)  |
|                | 0 No crack propagation   |
|                | 1 Fatigue type crack propagation   |
|                | 2 Direct crack propagation   |
| IGROW<br>(6-2) | How crack grows (Integer; Default = 2)   |
|                | 1 Uses remeshing (not presently available)   |

| Descriptor      | Meaning   |
|-----------------|---|
|                 | 2      Release user tyings or glued contact   |
| INCM<br>(6-3)   | Specifies the crack growth increment (Integer; Default = 2)   |
|                 | 1      Uses a fixed increment or a user subroutine  |
|                 | 2      Uses Paris law   |
| METHOD<br>(6-4) | Specifies the method used for the estimated crack growth direction (Integer; Default=1)   |
|                 | 1      Maximum hoop stress criterion (Default)  |
| TIME<br>(6-5)   | Time period for fatigue load sequence. (Real; no Default) Only enter if ITYPE=1   |
| IACT<br>(3-1)   | Flag for activating or deactivating an existing crack (Integer; Default = 0)  |
|                 | 0      Leave as is  |
|                 | 1      Activate   |
|                 | 2      Deactivate   |
| CGI<br>(7-1)    | Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law.                   |
| GC<br>(7-2)     | Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9. |
| GTH<br>(7-3)    | Paris law energy release rate threshold. (Real; Default = 0.0)  |
| C<br>(7-4)      | Paris law parameter C. (Real; Default = 0.0) Only enter if INCM=2   |
| M<br>(7-5)      | Paris law parameter m. (Real; Default = 0.0) Only enter if INCM=2   |
| GMIN<br>(7-6)   | Minimum growth increment. (Real; Default = 0.0) Only enter if INCM=2  |
| GC-II (7-4)     | Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.  |
| GC-III          | Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.   |
| TABCGI<br>(8-1) | TABLEMi or TABL3Di for CGI (crack growth increment). (Integer; Default = 0)   |
| TABGC<br>(8-2)  | TABLEMi or TABL3Di for scaling CG (fracture toughness). (Integer; Default = 0)  |

| Descriptor       | Meaning  |
|------------------|--|
| TABGTH<br>(8-3)  | TABLEMi or TABL3Di for scaling GTH (Paris law energy release rate) (Integer; Default = 0)  |
| TABC<br>(8-4)    | TABLEMi or TABL3Di for scaling C (Paris law parameter C). (Integer; Default = 0)   |
| TABM<br>(8-5)    | TABLEMi or TABL3Di for scaling M (Paris law parameter m). (Integer; Default = 0)   |
| TABGMIN<br>(8-6) | TABLEMi or TABL3Di for scaling GMIN (Minimum growth increment). (Integer; Default = 0)   |
| TAGC-II          | TABLEMi or TABL3Di for scaling GC-II (Integer; Default = 0)  |
| TAGC-III         | TABLEMi or TABL3Di for scaling GC-III (Integer; Default = 0)   |
| Gi<br>(5-i)      | Grids along the crack front – for a shell there is only one node. (Integer; no Default; at least one value, G1, must be provided. See Remark 6.) |

### Remarks:

1. This entry corresponds to Marc's VCCT model definition and history definition options.
2. (i,j) corresponds to Marc Vol C VCCT entry ith datablock jth field
3. ID corresponds to a Case Control VCCT command. Set ID=0 to enter VCCT entries into Marc's model definition.
4. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
5. This entry may be repeated as many times as necessary to describe all the cracks in the model.
6. The 4<sup>th</sup> line may be repeated as many times as necessary to describe all grids on the crack front
7. If the Alternate Format is used, entries may only be made in the fields indicated, however this line may be repeated as many times as necessary to describe all grids on the crack front.
8. If G1 is negative, the absolute value of G1 is used as the ID of a SET3 entry providing the list of grids. For this case, G2, G3, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
9. Most analyses do not use all of the entries on the second line, however a meaningful analysis will specify one or more of these values. Those not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
10. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

**VIEW****View Factor Definition**

Defines radiation cavity and shadowing for radiation view factor calculations.

**Format:**

| 1    | 2     | 3       | 4     | 5  | 6  | 7      | 8 | 9 | 10 |
|------|-------|---------|-------|----|----|--------|---|---|----|
| VIEW | IVIEW | ICAVITY | SHADE | NB | NG | DISLIN |   |   |    |

**Example:**

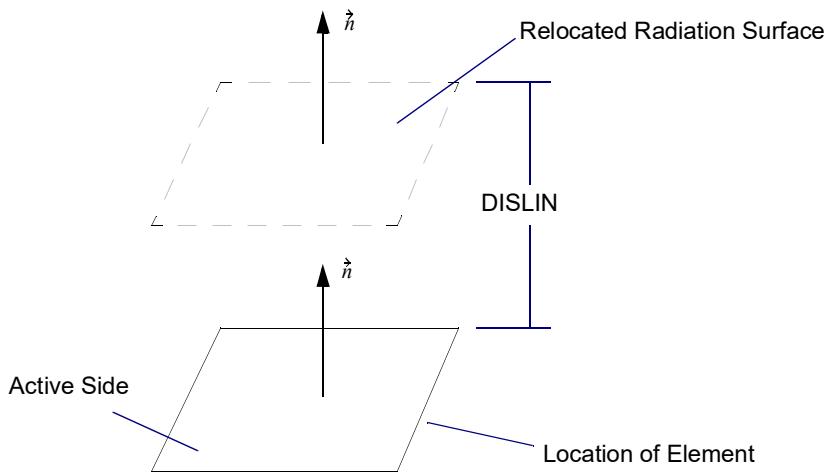
|      |   |   |      |   |   |      |  |  |  |
|------|---|---|------|---|---|------|--|--|--|
| VIEW | 1 | 1 | BOTH | 2 | 3 | 0.25 |  |  |  |
|------|---|---|------|---|---|------|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| IVIEW      | Identification number. (Integer > 0)  |
| ICAVITY    | Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)  |
| SHADE      | Shadowing flag for the face of CHBDYi element. (Character; Default = "BOTH")<br>NONE means the face can neither shade nor be shaded by other faces.<br>KSHD means the face can shade other faces.<br>KBSHD means the face can be shaded by other faces.<br>BOTH means the face can both shade and be shaded by other faces. (Default) |
| NB         | Subelement mesh size in the beta direction. (Integer > 0; Default = 1)  |
| NG         | Subelement mesh size in the gamma direction. (Integer > 0; Default = 1)   |
| DISLIN     | The displacement of a surface perpendicular to the surface. See <a href="#">Figure 9-168</a> . (Real; Default = 0.0)  |

**Remarks:**

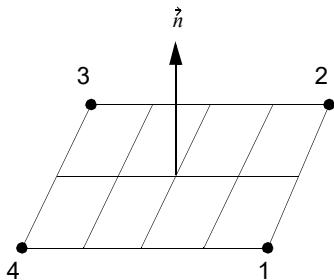
1. VIEW must be referenced by CHBDYE, CHBDYG, or CHBDYP elements to be used.
2. ICAVITY references the cavity to which the face of the CHBDYi element belongs; a zero or blank value indicates this face does not participate in a cavity.
3. NB, NG, and DISLIN are used in the calculation of view factors by finite difference or contour integration techniques. They are not used with the VIEW3D entry.
4. A summary of the shadowing conditions can be requested by the PARAM,MESH,YES Bulk Data entry.
5. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.

6. DISLIN should only be used with LINE type CHBDYE and CHBDYP surface elements. DISLIN > 0.0 means into the cavity. See [Figure 9-168](#).



**Figure 9-168** DISLIN Convention

7. NB and NG define the subelement mesh refinement when using the VIEW module (as opposed to the VIEW3D module) for the calculation of view factors.



**Figure 9-169** Typical AREA4 surface element where NB=2 and NG=4

8. For RC network solver in thermal analysis, the SHADE, NB, NG and DISLIN are ignored.

**VIEW3D****View Factor Definition - Gaussian Integration Method**

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

**Format:**

| 1      | 2       | 3    | 4    | 5    | 6    | 7    | 8    | 9      | 10 |
|--------|---------|------|------|------|------|------|------|--------|----|
| VIEW3D | ICAVITY | GITB | GIPS | CIER | ETOL | ZTOL | WTOL | RADCHK |    |

**Example:**

|        |   |   |   |   |  |        |  |  |  |
|--------|---|---|---|---|--|--------|--|--|--|
| VIEW3D | 1 | 2 | 2 | 4 |  | 1.0E-6 |  |  |  |
|--------|---|---|---|---|--|--------|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| ICAVITY    | Radiant cavity identification number on RADCAV entry. (Integer > 0)  |
| GITB       | Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)   |
| GIPS       | Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)   |
| CIER       | Discretization level used in the semi-analytic contour integration method. (1 ≤ Integer ≤ 20; Default = 4)   |
| ETOL       | Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real ≥ 0.0; Default = 0.1)   |
| ZTOL       | Assumed level of calculation below which the numbers are considered to be zero. (Real ≥ 0.0; Default = 1.E-10)   |
| WTOL       | Assumed degree of warpage above which the actual value of $F_{ii}$ will be calculated. (0.0 ≤ Real ≤ 1.0; Default = 0.01)  |
| RADCHK     | Type of diagnostic output desired for the radiation exchange surfaces. (Integer; Default = 3) <ul style="list-style-type: none"> <li>-1 No diagnostic output requested</li> <li>1 Grid table and element connectivity</li> <li>2 Surface Diagnostics - Surface type, area, skewness, taper, warpage, grid point sequencing, aspect ratio, and shading flags.</li> <li>3 Area, view factor, area-view factor product with error estimate, existence flags for partial self-shadowing, third-body shadowing with error estimate, and enclosure summations for view factor. (Default)</li> <li>0 Same as RADCHK = 1, 2, and 3</li> <li>12 Same as RADCHK = 1 and 2</li> </ul> |

| Descriptor | Meaning                  |
|------------|--------------------------|
| 13         | Same as RADCHK = 1 and 3 |
| 23         | Same as RADCHK = 2 and 3 |

**Remarks:**

1. For ETOL, when the error estimate exceeds the value input for the ETOL entry, the contour method is employed to develop an improved view factor.
2. For ZTOL, the use of a geometry scale that results in small numerical values of  $A_i F_{ij}$  should be avoided.
3. When WTOL is exceeded, the actual value of  $F_{ii}$  will be calculated when using the adaptive view module. Warpage will not be considered in the calculation of  $F_{ij}$ .
4. For axisymmetric analysis, RADCHK = -1 or 3 only.

**VIEWEX**

## Advanced RC Radiation Solver Parameters - SOL 400

Defines the radiation solver and correlating solver parameters for radiation calculations in RC heat transfer.  
 Note: You must have a copy of the external radiation code to use it.

## Format: (NEVADA)

| 1      | 2        | 3                    | 4                        | 5                 | 6                             | 7                | 8          | 9               | 10 |
|--------|----------|----------------------|--------------------------|-------------------|-------------------------------|------------------|------------|-----------------|----|
| VIEWEX | ICAVITY  | Run<br>Interactively | RADK<br>Distro<br>Method | Orbital           | Re-use<br>existing<br>results |                  |            |                 |    |
|        | "NEVADA" | RENO<br>Reflection   | Restart                  | Reno Ray<br>count | Vegas Ray<br>count            | Energy<br>Cutoff | Confidence | GRID<br>closure |    |
|        |          | GRID<br>iterations   | Time Scale               | RADK cutoff       |                               |                  |            |                 |    |

## Example:

|        |        |     |      |       |      |    |     |       |  |
|--------|--------|-----|------|-------|------|----|-----|-------|--|
| VIEWEX | 2      | T   | FULL | T     | F    |    |     |       |  |
|        | NEVADA | T   | T    | 5000  | 5000 | -3 | 99. | 0.001 |  |
|        |        | 300 | 1.0  | 1.0-8 |      |    |     |       |  |

## Format: (TSS)

| 1      | 2       | 3                    | 4                        | 5       | 6                             | 7 | 8 | 9 | 10 |
|--------|---------|----------------------|--------------------------|---------|-------------------------------|---|---|---|----|
| VIEWEX | ICAVITY | Run<br>Interactively | RADK<br>Distro<br>Method | Orbital | Re-use<br>existing<br>results |   |   |   |    |
|        | "TSS"   |                      |                          |         |                               |   |   |   |    |

## Example:

|        |     |   |      |   |   |  |  |  |  |
|--------|-----|---|------|---|---|--|--|--|--|
| VIEWEX | 3   | T | FULL | T | F |  |  |  |  |
|        | TSS |   |      |   |   |  |  |  |  |

## Format: (THERMICA)

| 1      | 2              | 3                    | 4                        | 5                   | 6                             | 7                           | 8                      | 9                         | 10 |
|--------|----------------|----------------------|--------------------------|---------------------|-------------------------------|-----------------------------|------------------------|---------------------------|----|
| VIEWEX | ICAVITY        | Run<br>Interactively | RADK<br>Distro<br>Method | Orbital             | Re-use<br>existing<br>results |                             |                        |                           |    |
|        | "THERMIC<br>A" | Solar Flux           | Planet<br>Albedo         | Planet<br>BlackBody | Restart                       | Suppress VF<br>Articulation | Radiation ray<br>count | Orbital flux<br>ray count |    |
|        |                | Confidence           | Time scale               | RADK cutoff         |                               |                             |                        |                           |    |

## Example:

|        |   |   |      |   |   |  |  |  |  |
|--------|---|---|------|---|---|--|--|--|--|
| VIEWEX | 4 | T | FULL | T | F |  |  |  |  |
|--------|---|---|------|---|---|--|--|--|--|

|  |          |        |     |       |   |   |      |      |  |
|--|----------|--------|-----|-------|---|---|------|------|--|
|  | THERMICA | 1380.0 | 0.3 | -19.0 | T | F | 5000 | 5000 |  |
|  |          | 99.0   | 1.0 | 1.0-8 |   |   |      |      |  |

Format: (TRASYS)

| 1      | 2        | 3                 | 4                  | 5                | 6                       | 7           | 8 | 9 | 10 |
|--------|----------|-------------------|--------------------|------------------|-------------------------|-------------|---|---|----|
| VIEWEX | ICAVITY  | Run Interactively | RADK Distro Method | Orbital          | Re-use existing results |             |   |   |    |
|        | "TRASYS" | Axi Radial mesh   | Axi Axial mesh     | Axi Angular mesh | Time scale              | RADK cutoff |   |   |    |

Example:

|        |        |   |      |   |     |       |  |  |  |
|--------|--------|---|------|---|-----|-------|--|--|--|
| VIEWEX | 5      | T | FULL | T | F   |       |  |  |  |
|        | TRASYS | 1 | 1    | 4 | 1.0 | 1.0-8 |  |  |  |

Format: (SRR)

| 1      | 2       | 3                    | 4                  | 5                 | 6                       | 7                 | 8                 | 9               | 10 |
|--------|---------|----------------------|--------------------|-------------------|-------------------------|-------------------|-------------------|-----------------|----|
| VIEWEX | ICAVITY | Run Interactively    | RADK Distro Method | Orbital           | Re-use existing results |                   |                   |                 |    |
|        | "SRR"   | Gebhart Solver       | Convergence Tol    | Max Iter          | Fij smoothing method    | Fij Filter cutoff | Fij Smoothing Tol | Fij Smooth Iter |    |
|        |         | Bij smoothing method | Bij Filter cutoff  | Bij Smoothing Tol | Bij Max Iter            |                   |                   |                 |    |

Example:

|        |     |      |       |       |      |       |       |    |  |
|--------|-----|------|-------|-------|------|-------|-------|----|--|
| VIEWEX | 6   | T    | FULL  | T     | F    |       |       |    |  |
|        | SRR | GS   | 1.0-5 | 50    | CROP | 1.0-8 | 1.0-4 | 50 |  |
|        |     | CROP | 1.0-8 | 1.0-4 | 50   |       |       |    |  |

Format: (SRQ)

| 1      | 2       | 3                 | 4                  | 5        | 6                       | 7             | 8                 | 9               | 10 |
|--------|---------|-------------------|--------------------|----------|-------------------------|---------------|-------------------|-----------------|----|
| VIEWEX | ICAVITY | Run Interactively | RADK Distro Method | Orbital  | Re-use existing results |               |                   |                 |    |
|        | "SRQ"   | Flux solver       | Convergence Tol    | Max Iter | Fij smoothing method    | Filter cutoff | Fij Smoothing Tol | Fij Smooth Iter |    |

Example:

|        |     |    |       |    |      |       |       |    |  |
|--------|-----|----|-------|----|------|-------|-------|----|--|
| VIEWEX | 7   | T  | FULL  | T  | F    |       |       |    |  |
|        | SRQ | GS | 1.0-5 | 50 | CROP | 1.0-8 | 1.0-4 | 50 |  |

| Descriptor               | Meaning  |
|--------------------------|--|
| ICAVITY                  | Cavity identification number (Integer > 0; Required)   |
| Run Interactively        | Run the radiation code interactively (Character; "T" or "F", Default "F")<br>Do not currently have batch mode for Thermica or TSS<br>Do not have interactive mode for TRASYS |
| RADK Distro Method       | How to redistribute RADK onto elements. (Character; "FULL", "AREA", or "DIRECT"; Default "FULL")   |
| Orbital                  | Use orbital analysis for radiation (Character; "T" or "F"; Default "F")<br>Not supported in SindaRad   |
| Re-use existing results  | Re-use previous radiation results (Character; "T" or "F"; Default "F")   |
| "NEVADA"                 | Identification that NEVADA will be used (Character)  |
| RENO Reflection          | Use reflection method of ray tracing for RADK (Character; "T" or "F"; Default "T")   |
| Restart                  | Use Restart (Character; "T" or "F"; Default "F")   |
| Reno Ray count           | Number of rays cast for Reno module (Integer > 0; Default 5000)  |
| Vegas Ray count          | Number of rays cast for Vegas module (Integer > 0; Default 5000)   |
| Energy Cutoff            | Energy cutoff level (Integer; Default -3)  |
| Confidence               | Confidence Level % (Real > 0.0; Default 99.0)  |
| GRID closure             | GRID closure tolerance (Real > 0.0; Default 0.001)   |
| GRID iterations          | Maximum GRID iterations (Integer > 0; Default 300)   |
| Time Scale               | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)   |
| RADK cutoff              | RADK filter smallest element (Real $\geq$ 0.0; Default 1.0e-8)   |
| "TSS"                    | Identification that TSS will be used (Character)   |
| "THERMICA"               | Identification that THERMICA will be used (Character)  |
| Solar Flux               | Quantity of solar flux (Real > 0.0; Default 1380.0 W/m <sup>2</sup> )  |
| Planet Albedo            | Planetary Albedo (Real; Default 0.3; assumes Earth orbit)  |
| Planet BlackBody         | Planet Blackbody (Real; Default -19.0; assumes Earth orbit)  |
| Restart                  | Use Restart option (Character, "T" or "F"; Default "F")  |
| Suppress VF Articulation | Suppress view factor articulation (Character; "T" or "F"; Default "T")   |
| Radiation ray count      | Number of rays cast for radiation calculation (Integer > 0; Default 5000)  |
| Orbital flux ray count   | Number of rays cast for orbital flux (Integer > 0; Default 5000)   |
| Confidence               | Confidence Level % (Real > 0.0; Default 99.0)  |
| Time Scale               | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)   |

| Descriptor           | Meaning   |
|----------------------|---|
| RADK cutoff          | RADK filter smallest element (Real $\geq 0.0$ ; Default 1.0e-8)   |
| “TRASYS”             | Identification that TRASYS will be used (Character)   |
| Axi Radial mesh      | Axisymmetric element mesh in radial direction (Integer $> 0$ ; Default 1)   |
| Axi Axial mesh       | Axisymmetric element mesh in axial direction (Integer $> 0$ ; Default 1)  |
| Axi Angular mesh     | Axisymmetric element mesh in angular direction (Integer $> 0$ ; Default 4)  |
| Time Scale           | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real $> 0.0$ ; Default 1.0) |
| RADK cutoff          | RADK filter smallest element (Real $\geq 0.0$ ; Default 1.0e-8)   |
| “SRR”                | Identification that the SindaRad RADK method will be used (Character)   |
| Gebhart Solver       | Which RADK solver to use (Character; “GS” or “FGS”; Default “GS”)   |
| Convergence Tol      | Tolerance for convergence of RADK calculation (Real $\geq 0.0$ ; Default 1.0e-5)  |
| Max Iter             | Maximum allowable iterations to converge (Integer $> 0$ ; Default 50)   |
| Fij smoothing method | How to filter view factors (Character; “CROP” or “HIGH”; Default “CROP”)  |
| Fij Filter cutoff    | Parameter for filter (Real $> 0.0$ ; Default 1.0e-8)  |
| Fij Smoothing Tol    | Tolerance for smoothing (Real $\geq 0$ ; Default 1.0e-4)  |
| Fij Smooth Iter      | Maximum allowable iterations to smoothing (Integer $> 0$ ; Default 50)  |
| Bij smoothing method | How to filter conductors (Character; “CROP” or “HIGH”; Default “CROP”)  |
| Bij Filter cutoff    | Parameter for filter (Real $> 0.0$ ; Default 1.0e-8)  |
| Bij Smoothing Tol    | Tolerance for smoothing (Real $\geq 0$ ; Default 1.0e-4)  |
| Bij Max Iter         | Maximum allowable iterations to smoothing (Integer $> 0$ ; Default 50)  |
| “SRQ”                | Identification that the SindaRad QRad method will be used (Character)   |
| Flux Solver          | Which QRad solver to use (Character, “GS” or “CG”; Default “GS”)  |
| Convergence Tol      | Tolerance for convergence of QRad calculation (Real $\geq 0.0$ ; Default 1.0e-5)  |
| Max Iter             | Maximum allowable iterations to converge (Integer $> 0$ ; Default 50)   |
| Fij smoothing method | How to filter view factors (Character, “CROP” or “HIGH”; Default “CROP”)  |
| Fij Filter cutoff    | Parameter for filter (Real $> 0.0$ ; Default 1.0e-8)  |
| Fij Smoothing Tol    | Tolerance for smoothing (Real $\geq 0$ ; Default 1.0e-4)  |
| Fij Smooth Iter      | Maximum allowable iterations to smoothing (Integer $> 0$ ; Default 50)  |

**Remarks:**

1. This entry is for RC Network solver only.
2. Each entry type is designed for one specific radiation solver, except the very last two types, which are for SindaRad’s two options.

NEVADA

TSS

TERMICA

TRASYS

SINDARad RADK method

SINDARad Q method

3. For more details about the parameters in the entry, please reference SINDA for *Patran User's Guide* and the *SINDARad User's Guide*.
4. MSC provides these radiation programs on Windows only.

**WALL****Rigid Wall**

Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Finite or infinite size (FINITE), Orthotropic friction (ORTHO) a mass and an initial velocity (MOVING) and Force output option (FORCES) can be defined. Used in SOL 700 only.

**Format:**

| 1    | 2      | 3  | 4  | 5   | 6  | 7  | 8  | 9      | 10 |
|------|--------|----|----|-----|----|----|----|--------|----|
| WALL | ID     | XP | YP | ZP  | NX | NY | NZ | BCGRID |    |
|      | METHOD | FS | FK | EXP |    |    |    |        |    |

**Example:**

|      |         |     |  |  |  |  |     |    |  |
|------|---------|-----|--|--|--|--|-----|----|--|
| WALL | 17      |     |  |  |  |  | 1.0 | 21 |  |
|      | PENALTY | 0.2 |  |  |  |  |     |    |  |

| Descriptor | Meaning   |   |
|------------|---|---|
| ID         | Unique rigid-wall number. (Integer > 0; Required)   |   |
| XP, YP, ZP | Coordinates of the origin of the wall. (Real; Default = 0.0)  |   |
| NX, NY, NZ | A vector normal to the wall pointing towards the model. (Real; Default = 0.0)                       |   |
| BCGRID     | Number of a BCGRID entry listing the points that cannot penetrate the wall. (Integer > 0; Required) |   |
| METHOD     | Algorithm for contact processing. (Character; Default = PENALTY)                                    |   |
|            | PENALTY   | Penalty method, allowing for extra boundary conditions, friction and output.  |
|            | KINMATIC  | Kinematic method, only included for compatibility reasons with older Dytran versions. This method allows no extra boundary conditions, no friction and no output. |
| FS         | Static coefficient of friction. See Remark 5. (Real $\geq 0$ ; Default = 0.0)                       |   |
| FK         | Kinetic coefficient of friction. See Remark 5. (Real $\geq 0$ ; Default = 0.0)                      |   |
| EXP        | Exponential decay coefficient. See Remark 5. (Real $\geq 0$ ; Default = 0.0)                        |   |

### Remarks

1. A rigid plane of infinite size is generated that the grid points cannot penetrate. The plane is fixed in space.
2. The grid points can slide on the wall and separate from it.
3. A (moving) rigid plane of finite size can be modeled by using a rigid surface and a master-slave contact.

4. For the wall definition using penalty method, output can be requested by referencing it in a SET command in the Case Control Section. The keywords for output are WALLS and WALLOUT. Please check \$S700.
5. The coefficient of friction is given by:

$$\mu = \mu_k + (\mu_s - \mu_k)e^{-\beta v}$$

where

$\mu_s$  = Static coefficient of friction FS.

$\mu_k$  = Kinetic coefficient of friction FK.

$\beta$  = Exponential decay coefficient EXP.

$v$  = Relative sliding velocity at the point of contact.

**WEAR****Specifies Values for Modeling Mechanical Wear in Deformable Contact Bodies**

Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.

**Format:**

| 1    | 2       | 3       | 4       | 5       | 6    | 7    | 8 | 9 | 10 |
|------|---------|---------|---------|---------|------|------|---|---|----|
| WEAR | BID     | Model   | IUPD    |         |      |      |   |   |    |
|      | COEF    | HARD    | SEXP    | VEXP    | B    | SFGU |   |   |    |
|      | T(COEF) | T(HARD) | T(SEXP) | T(VEXP) | T(B) |      |   |   |    |

**Example:**

|      |     |      |     |     |      |     |  |  |  |
|------|-----|------|-----|-----|------|-----|--|--|--|
| WEAR | 3   | 2    | 1   |     |      |     |  |  |  |
|      | 3.6 | 200. | 1.1 | 1.2 | .087 | 1.0 |  |  |  |
|      | 22  | 23   | 24  | 25  | 26   |     |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| BID        | BCBODY ID for which this wear entry applies. (Integer > 0; no Default; see Remark 2)   |
| MODEL      | Wear model. (Integer > 0; Default = 1, see Remark 1)   |
| IUPD       | Geometry update due to wear flag. (Integer; Default = 0) <ul style="list-style-type: none"> <li>0      Do not update the geometry</li> <li>1      Update the geometry</li> </ul> |
| COEF       | Wear coefficient. (Real; no Default)   |
| HARD       | Material hardness. (Real; no Default)  |
| SEXP       | Stress exponent. (Real; Default = 1.0)   |
| VEXP       | Velocity exponent. (Real; Default = 1.0)   |
| B          | Thermal activation constant. (Real; Default = 1.0)   |
| SFGU       | Scale factor for geometry update. (Real; Default = 1.0)  |
| T(COEF)    | TABL3D ID for the wear coefficient. (Integer; Default = 0, meaning no table variation)   |
| T(HARD)    | TABL3D ID for the material hardness. (Integer; Default = 0, meaning no table variation)  |
| T(SEXP)    | TABL3D ID for the stress exponent. (Integer; Default = 0; meaning no table variation)  |
| T(VEXP)    | TABL3D ID for the velocity exponent. (Integer; Default = 0, meaning no table variation)  |
| T(B)       | TABL3D ID for the thermal activation constant. (Integer; Default = 0, meaning no table variation)  |

**Remarks:**

1. The following wear models are available

$$\dot{w} = \frac{K}{H} \sigma v_{rel} \quad \text{Archard base model}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n \quad \text{Archard model; Bayer exponential form}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n e^{\frac{-B}{T}} \quad \text{Archard model; exponential form with thermal activation}$$

2. BID may also point the ID in field 2 of BCPROP, BCMATL or BCBOX.

**WETELME**

Wetted element in OpenFSI by side in SOL 400

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2    | 3   | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|-----|------|---|---|---|---|---|----|
| WETELME | WEID | EID | SIDE |   |   |   |   |   |    |

Example:

|         |       |    |   |  |  |  |  |  |  |
|---------|-------|----|---|--|--|--|--|--|--|
| WETELME | 10001 | 34 | 3 |  |  |  |  |  |  |
|---------|-------|----|---|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| WEID       | Wetted element identification number. (Integer > 0; no Default)   |
| EID        | Structural element identification number, which corresponds to a surface element CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR; or a solid element CTETRA, CPENTA, or CHEXA. (Integer > 0; no Default) |
| SIDE       | Side identification number of element EID. ( $1 \leq \text{Integer} \leq 6$ ; no Default)   |

**WETELMG**

## Wetted element in OpenFSI by type in SOL 400

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

**Format:**

| 1       | 2    | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|------|----|----|----|----|----|----|----|
| WETELMG | WEID | TYPE |    |    |    |    |    |    |    |
|         | G1   | G2   | G3 | G4 | G5 | G6 | G7 | G8 |    |

**Example:**

|         |       |       |     |    |  |  |  |  |  |
|---------|-------|-------|-----|----|--|--|--|--|--|
| WETELMG | 10001 | QUAD4 |     |    |  |  |  |  |  |
|         | 23    | 35    | 124 | 28 |  |  |  |  |  |

| Descriptor  | Meaning  |
|-------------|--|
| WEID        | Wetted element identification number. (Integer > 0; no Default)  |
| TYPE        | Wetted element type, which can be any of TRIA3, TRIA6, QUAD4, QUAD8, LINE2 or LINE3. (Character; no Default) |
| G1, ..., G8 | Grid point identification numbers for the wetted surface element WEID. (Integer > 0; no Default)             |

**WETLOAD**

Load set and associated wetted surface for OpenFSI in SOL 400

Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2    | 3    | 4       | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|------|---------|---|---|---|---|---|----|
| WETLOAD | WLID | WSID | SERV_ID |   |   |   |   |   |    |

Example:

|         |   |      |        |  |  |  |  |  |  |
|---------|---|------|--------|--|--|--|--|--|--|
| WETLOAD | 1 | 1001 | scafsi |  |  |  |  |  |  |
|---------|---|------|--------|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| WLID       | Load set ID, referenced by the EXCITEID field in the TLOAD entry. (Integer > 0; no Default)  |
| WSID       | Wetted surface identification number. The wetted surface must be defined in the WETSURF Bulk Data entry. (Integer > 0; no Default)                                 |
| SERV_ID    | OpenFSI SCA service name associated with the wetted surface loads. The OpenFSI SCA service is defined using the CONNECT SERVICE FMS entry. (Character; no Default) |

Remarks:

1. To activate WETLOAD, the following entries must be present and referenced:
  - a. The load set ID WLID must be referenced by a TLOAD1 entry.
  - b. The TLOAD1 entry must be referenced by a DLOAD Case Control command or a DLOAD Bulk Data entry, which in turn is referenced by a DLOAD Case Control command.
  - c. The TLOAD1 entry must reference a TABLED1 entry.
  - d. The TABLED1 entry must have constant values and be the same for all TLOAD1 entries referenced by the WETLOAD entries used within the same service.

Procedures a ~ d are required in the Nastran load generator processes. Their values do not really influence the load calculation of WETLOAD in OpenFSI. Please refer to the remarks below to apply WETLOAD(s) to OpenFSI.

2. The SERV\_ID string must be 8 characters or less.
3. For multiple WETLOAD entries with same SERV\_ID, the wet surfaces in WSID will be merged together for the service and the WLID in the first WETLOAD entry will replace other entries WLID.
4. In the current implementation, all OpenFSI services in a model are active and participate in the calculation. The TLOAD1 and DLOAD selection in case control are not applied on OpenFSI services.
5. For OpenFSI services, the scale factors in DLOAD are always taken to be constant 1.0.

6. In the FMS Section, the following statement should be included: CONNECT SERVICE SERV\_ID ExternalCodeVendor.OpenFSI

**WETSURF**

Wetted surface used in OpenFSI in SOL 400

Defines a wetted surface used in OpenFSI fluid structure interaction simulations in SOL 400 only.

**Format:**

| 1       | 2     | 3      | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|--------|--------|-------|-------|-------|-------|-------|----|
| WETSURF | WSID  | WTAG   |        |       |       |       |       |       |    |
|         | WEID1 | WEID2  | WEID3  | WEID4 | WEID5 | WEID6 | WEID7 | WEID8 |    |
|         | WEID9 | WEID10 | -etc.- |       |       |       |       |       |    |

**Alternate Format:**

|         |       |        |       |      |     |  |  |  |  |
|---------|-------|--------|-------|------|-----|--|--|--|--|
| WETSURF | WSID  | WTAG   |       |      |     |  |  |  |  |
|         | WEID1 | "THRU" | WEID2 | "BY" | INC |  |  |  |  |

**Example:**

|         |       |       |     |    |    |    |    |    |  |
|---------|-------|-------|-----|----|----|----|----|----|--|
| WETSURF | 10001 | wall1 |     |    |    |    |    |    |  |
|         | 5     | THRU  | 21  | BY | 4  |    |    |    |  |
|         | 27    | 30    | 33  |    |    |    |    |    |  |
|         | 35    | THRU  | 44  |    |    |    |    |    |  |
|         | 67    | 68    | 70  | 72 | 77 | 82 | 86 | 79 |  |
|         | 89    | THRU  | 110 | BY | 3  |    |    |    |  |

| Descriptor           | Meaning   |
|----------------------|---|
| WSID                 | Wetted surface identification number. (Integer > 0; no Default)   |
| WTAG                 | Wetted surface tag name exported to an external code using the OpenFSI SCA interface. (Character; no Default)           |
| WEID1,<br>WEID2, ... | Wetted element identification numbers defined using the WETELMG or WETELME Bulk Data entries. (Integer > 0; no Default) |
| THRU, BY             | Keywords to specify a range of wetted elements. (Character; no Default)   |
| INC                  | Increment to use with the "THRU" and "BY" keywords. (Integer; Default = 1)  |

## YLDHY

### Hydrodynamic Yield Model

Defines a yield model with zero yield stress. Used in SOL 700 only.

Format:

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|---|---|---|---|---|---|----|
| YLDHY | YID |   |   |   |   |   |   |   |    |

Example:

|       |     |  |  |  |  |  |  |  |  |
|-------|-----|--|--|--|--|--|--|--|--|
| YLDHY | 200 |  |  |  |  |  |  |  |  |
|-------|-----|--|--|--|--|--|--|--|--|

| Descriptor | Meaning   |
|------------|---|
| YID        | Unique yield-model number referenced from a MATDEUL entry. (Integer > 0;<br>Required) |

Remark:

1. This yield model should be used for fluids that have no shear strength.
2. YID must unique among all YLDxx entries in one model.

**YLDJC****Johnson-Cook Yield Model**

Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature. Used in SOL 700 only.

Format:

| 1     | 2   | 3     | 4     | 5   | 6   | 7   | 8    | 9    | 10 |
|-------|-----|-------|-------|-----|-----|-----|------|------|----|
| YLDJC | YID | A     | B     | n   | C   | m   | EPS0 | CP   |    |
|       | 100 | 200E6 | 50.E6 | 0.1 | .15 | .95 | 1.   | 285. |    |

Example:

|       |       |       |  |  |  |  |  |  |  |
|-------|-------|-------|--|--|--|--|--|--|--|
| YLDJC | TMELT | TROOM |  |  |  |  |  |  |  |
|       | 1500. | 273.  |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield-model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A          | Static yield stress. (Real $\geq 0.0$ ; Required)                                  |
| B          | Hardening parameter. (Real; Default = 0.0)   |
| n          | Hardening exponent. (Real; Default = 1.0)  |
| C          | Strain-rate parameter. (Real; Default = 0.0)                                       |
| m          | Temperature exponent. (Real; Default = 1.0)  |
| EPS0       | Reference strain rate. (Real $> 0.0$ ; Default = 1.0)                              |
| CP         | Specific heat. (Real $> 0.0$ ; Default = 1.E20)                                    |
| TMELT      | Melt temperature. (Real; Default = 1.E20)  |
| TROOM      | Room temperature. (Real; Default = 293.0)  |

Remark:

1. The yield stress is computed from

$$\sigma_y = (A + B\dot{\varepsilon}_p^n) \left( 1 + C \ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right) (1 - T^*^m) \right)$$

where

$$\dot{\varepsilon}_p = \text{effective plastic strain}$$

$$T^* = \frac{(T - T_r)}{(T_m - T_r)}$$

|                    |   |                        |
|--------------------|---|------------------------|
| $\dot{\epsilon}$   | = | effective strain rate  |
| $\dot{\epsilon}_0$ | = | referenced strain rate |
| $T$                | = | temperature            |
| $T_r$              | = | room temperature       |
| $T_m$              | = | melt temperature       |

and  $A$ ,  $B$ ,  $n$ ,  $C$ , and  $m$  are constants.

2. The reference strain rate is per unit time.
3. YID must unique among all YLDxx entries in one model.

**YLDMC****Mohr-Coulomb Yield Model**

Defines a Mohr-Coulomb yield model. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----|----|----|---|---|---|---|----|
| YLDMC | YID | Y1 | Y2 | Y3 |   |   |   |   |    |

**Example:**

|       |   |       |       |      |  |  |  |  |  |
|-------|---|-------|-------|------|--|--|--|--|--|
| YLDMC | 1 | 10.E5 | 20.E5 | 1.E4 |  |  |  |  |  |
|-------|---|-------|-------|------|--|--|--|--|--|

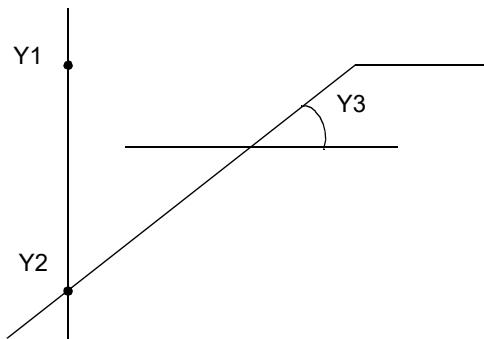
| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield-model number referenced from MATDEUL for Eulerian elements with shear strength. (Integer > 0; Required) |
| Y1         | Cap yield stress. (Real; Required)   |
| Y2         | Cohesion. (Real; Required)   |
| Y3         | Internal friction angle. (Real; Required)  |

**Remarks:**

1. The yield stress depends on the pressure as

$$\sigma_y = \text{MIN}(Y1, (Y2 + Y3 \cdot P))$$

where Y1, Y2, Y3 are constants and P is the pressure.



2. This yield model is applicable only for Eulerian materials with shear strength.
3. YID must unique among all YLDxx entries in one model.

**YLDMSS****Multi-Surface Yield Model for Snow**

Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3     | 4  | 5    | 6    | 7   | 8    | 9   | 10 |
|--------|-----|-------|----|------|------|-----|------|-----|----|
| YLDMSS | YID | KC    | T  | CC   | AC   | BC  | FC   | FTU |    |
|        | 7   | 0.149 | 82 | 1E-5 | 0.09 | 0.2 | 0.99 | 82  |    |

**Example:**

|        |       |     |  |  |  |  |  |  |  |
|--------|-------|-----|--|--|--|--|--|--|--|
| YLDMSS | ALP0  | DS  |  |  |  |  |  |  |  |
|        | -0.37 | 0.0 |  |  |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required)   |
| KC         | Parameter related to the angle of friction. (Real > 0; Required)                     |
| T          | Equivalent value of the snow cohesion. see Remark 5. (Real > 0; Required)            |
| CC         | Shape of the yield surface. See Remark 4. (Real > 0; Required)                       |
| AC         | Hardening parameter for compression. See Remark 4. (Real > 0; Required)              |
| BC         | Hardening parameter for compression. See Remark 4. (Real > 0; Required)              |
| FC         | Factor to avoid singularity. See Remark 4. (0 < Real < 1; Default = 0.99)            |
| FTU        | Hydrostatic tensile strength. See Remark 6. (Real > 0; Default = T/3)                |
| ALP0       | Initial compressive volumetric plasticity strain. See Remark 4. (Real < 0; Required) |
| DS         | Softening modulus. See Remark. (Real $\geq$ 0; Default = 0.0)                        |

**Remarks:**

1. This material model can be used to model Snow material.
2. In addition to deviatoric plastic strain there is also volumetric plastic strain. This volumetric strain is stored in the output variable VOLPLS. The deviatoric strain is stored in the variable EFFPLS.
3. For detail description of each parameter in this model, the user should refer to the Theory Manual in which the mechanical properties of snow are described.
4. If CC is set to zero then the material behaves as a Drucker-Prager model. Ac, Bc, Fc<sup>\*</sup> and ALP0 will be ignored.

5. The T value must be consistently converted from the cohesion data,  $C_{DP}$ , of the Drucker-Prager model as follows:

$$T = C_{DP}/K_C$$

6. FTU, hydrostatic tensile strength, may not be greater than T divided by 3. Otherwise it will be set to that value.
7. The softening modulus is used to update the hardening parameter  $q_i$ , see Theory Manual. It can be requested as output using FTU variable. The corresponding accumulated-plastic-volumetric-tensile-strain variable is SOFTE.
8. This material model is valid for the Euler with Strength solver and the Multi-material Euler with Strength solver.
9. YID must unique among all YLDxx entries in one model.

**YLDPOL****Polynomial Yield Model**

Defines a polynomial yield model where the yield stress is a function of effective plastic strain. Used in SOL 700 only.

Format:

| 1      | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9    | 10 |
|--------|-----|---|---|---|---|---|---|------|----|
| YLDPOL | YID | A | B | C | D | E | F | SMAX |    |

Example:

|        |   |        |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|
| YLDPOL | 7 | 180.E6 |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A          | Initial yield parameters. (Real > 0; Required)                             |
| B          | Coefficient B. (Real; Default = 0.0)                                       |
| C          | Coefficient C. (Real; Default = 0.0)                                       |
| D          | Coefficient D. (Real; Default = 0.0)                                       |
| E          | Coefficient E. (Real; Default = 0.0)                                       |
| F          | Coefficient F. (Real; Default = 0.0)                                       |
| SMAX       | Maximum yield stress. (Real; Default = 1.E20)                              |

Remarks:

1. The yield stress is computed from

$$\sigma_y = \text{MIN}(\sigma_{\max}, A + B\varepsilon_p + C\varepsilon_p^2 + D\varepsilon_p^3 + E\varepsilon_p^4 + F\varepsilon_p^5)$$

where

$\varepsilon_p$  = effective plastic strain

$\sigma_{\max}$  = maximum yield stress

and A, B, C, D, E and F are constants.

2. YID must unique among all YLDxx entries in one model.

**YLDRPL****Rate Power Law Yield Model**

Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate. Used in SOL 700 only.

Format:

| 1      | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|---|---|---|---|---|---|----|
| YLDRPL | YID | A | B | n | m | C |   |   |    |

Example:

|        |   |        |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|
| YLDRPL | 7 | 180.E6 |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A          | Initial yield parameters. (Real > 0; Required)                             |
| B          | Hardening parameter. (Real; Default = 0.0)                                 |
| N          | Hardening exponent. (Real; Default = 1.0)                                  |
| M          | Strain rate exponent. (Real; Default = 1.0)                                |
| C          | Minimum yield stress. (Real; Default = 1.E20)                              |

Remarks:

1. The yield stress is computed from

$$\sigma_y = \text{MAX}(C, A + B \varepsilon_p^{n \cdot m})$$

where

$\varepsilon_p$  = effective plastic strain

$\dot{\varepsilon}$  = effective strain rate

and  $A$ ,  $B$ ,  $n$ ,  $m$  and  $C$  are constants.

2. YID must unique among all YLDxx entries in one model.

## YLD SG

## Steinberg-Guinan Yield Model

Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature. Used in SOL 700 only.

Format:

| 1      | 2      | 3      | 4  | 5  | 6  | 7 | 8 | 9  | 10 |
|--------|--------|--------|----|----|----|---|---|----|----|
| YLD SG | YID    | A1     | A2 | A3 | A4 | H | B | CP |    |
|        | T MELT | T ROOM |    |    |    |   |   |    |    |

Example:

|        |      |      |        |     |     |  |  |  |  |
|--------|------|------|--------|-----|-----|--|--|--|--|
| YLD SG | 7    | 8E+6 | 100E+6 | 110 | 0.5 |  |  |  |  |
|        | 1500 | 273  |        |     |     |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A1-A4      | Yield parameters. (Real > 0; Required)   |
| H, B       | Yield parameters. See Remark 4. (Real; Default = 0.0)                              |
| CP         | Specific heat. (Real > 0; Default = 1.E20)   |
| T MELT     | Melt temperature. (Real; Default = 1.E20)  |
| T ROOM     | Room temperature. (Real; Default = 293.0)  |

Remarks:

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$A_T = A_1(1 + A_3 \varepsilon_p)^A$$

$$\sigma_y = \min(A_2, A_T) \left[ 1 - H(T - T_r) + B p \left( \frac{\rho}{\rho_{ref}} \right)^{\frac{1}{3}} \right] \quad T < T_m$$

$$\sigma_y = 0, \quad T \geq T_m$$

And  $A_1, \dots, A_4, H$  and  $B$  are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.

**YLDTM****Tanimura-Mimura Yield Model**

Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

**Format:**

| 1     | 2     | 3     | 4   | 5 | 6 | 7    | 8    | 9  | 10 |
|-------|-------|-------|-----|---|---|------|------|----|----|
| YLDTM | YID   | A     | B   | C | D | M    | EPSM | CP |    |
|       | TMELT | TROOM | SCR | E | K | EPS0 |      |    |    |

**Example:**

|       |   |        |         |       |     |     |       |  |  |
|-------|---|--------|---------|-------|-----|-----|-------|--|--|
| YLDTM | 7 | 45.6E6 | 19.5E6  |       |     | 1.0 | 0.001 |  |  |
|       |   |        | 4000.e6 | 2100. | 0.5 | 1.0 |       |  |  |

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A          | Static yield parameter. (Real > 0; Required)                               |
| B          | Hardening parameter. (Real; Default = 0.0)                                 |
| C          | Strain rate parameter C. (Real; Default = 0.0)                             |
| D          | Strain rate parameter D. (Real; Default = 0.0)                             |
| M          | Temperature exponent. (Real; Default = 0.0)                                |
| EPSM       | Quasi-static strain rate. (Real > 0; Default = 1.0)                        |
| CP         | Specific heat. (Real > 0; Default = 1.E20)                                 |
| TMELT      | Melt temperature. (Real; Default = 1.E20)                                  |
| TROOM      | Room temperature. (Real; Default = 293.0)                                  |
| SCR        | Critical yield stress. (Real > 0; Default = 1.0)                           |
| E          | Strain rate parameter E. (Real; Default = 0.0)                             |
| K          | Strain rate exponent. (Real; Default = 1.0)                                |
| EPS0       | Reference strain rate. (Real > 0; Default = 1.0)                           |

**Remarks:**

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$\sigma_Y = \left[ A + B\varepsilon_P + (C + D\varepsilon_P) \left( 1 - \frac{A + B\varepsilon_P}{\sigma_{cr}} \right) \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_s} \right) \right] (1 - T^m) + E \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right)^k$$

where

|                       |   |                          |
|-----------------------|---|--------------------------|
| $\varepsilon_p$       | = | effective plastic strain |
| $\sigma_{cr}$         | = | critical yield stress    |
| $\dot{\varepsilon}$   | = | effective strain rate    |
| $\dot{\varepsilon}_s$ | = | quasi-static strain rate |
| $\dot{\varepsilon}_0$ | = | reference strain rate    |
| $T^*$                 | = | $(T - T_r)/(T_m - T_r)$  |
| T                     | = | temperature              |
| $T_r$                 | = | room temperature         |
| $T_m$                 | = | melt temperature         |

and  $A, B, C, D, m, E$  and  $k$  are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.

**YLDUDS**

User-defined Yield Model for Elements.

Specifies that a user subroutine is being used to define a simple yield model. Use in SOL700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|---|---|---|---|---|----|
| YLDUDS | YLD | GROUP | UNAME |   |   |   |   |   |    |

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myyld 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| YLDUDS | 12 | myyld | EXYLD |   |   |   |   |   |    |

| Descriptor | Meaning   |
|------------|---|
| YLD        | Unique yield model ID. (Integer > 0; Required)  |
| GROUP      | The group name used for the FMS section CONNECT SERVICE statement.<br>(Character; no Default) |
| UNAME      | User subroutine name associated with the entry. (Character; default=EXYLD)                    |

**Remarks:**

1. Only can be used for SOL 700.
2. The YLD must be referenced by a MATDEUL or MAT1 entry.
3. This yield model is applicable only for Euler elements with shear strength
4. UNAME can be:

| Subroutine Name | Function                          |
|-----------------|-----------------------------------|
| EXYLD           | Standard user defined yield model |

5. The yield strength can depend on the amount of failure or damage of the Euler element and on the VOLPLS and SOFTE element variables. This amount of damage can be either specified by the damage variable of the FAILJC entry or by a more general failure estimate by using the FAILUDS entry with UNAME=EXFAIL2. The VOLPLS and SOFTE variables can also be filled by the YLDUDS entry and FAILUDS entry with UNAME=EXFAIL2.

6. The damage variable is determined by either FAILJC or FAILUDS entry with UNAME=EXFAIL2. In UNAME=EXYLD, the yield stress can be reduced depending on the magnitude of the damage variable. In that case the NOFAIL option should be set on the FAILJC or FAILUDS entry with UNAME=EXFAIL2.

**YLDVM**

## von Mises Yield Model

Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion. Used in SOL 700 only.

Format:

| 1     | 2     | 3     | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-------|-------|------|---|---|---|---|---|----|
| YLDVM | YID   | YIELD | EH   |   |   |   |   |   |    |
|       | TABLE | TYPE  | TABY | D | P |   |   |   |    |

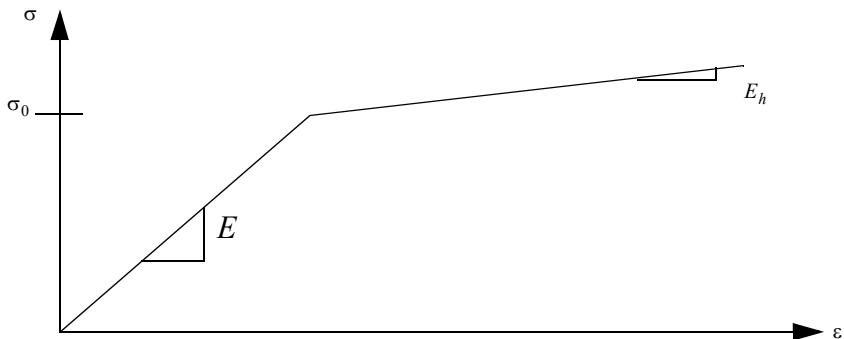
Example:

|       |    |        |         |  |  |  |  |  |  |
|-------|----|--------|---------|--|--|--|--|--|--|
| YLDVM | 32 | 250.E6 | 2000.E6 |  |  |  |  |  |  |
|       |    |        |         |  |  |  |  |  |  |

| Descriptor | Meaning  |                                  |
|------------|--|----------------------------------|
| YID        | Unique yield-model number referenced from MATDEUL. (Integer > 0; Required)   |                                  |
| YIELD      | Yield stress. (Real; Required)   |                                  |
| EH         | Hardening modulus. (Real; Default = 0.0)   |                                  |
| TABLE      | Number of a TABLED1 entry giving the variation of effective stress (y-value) with effective strain (x-value). See Remark 4. (Integer > 0)  |                                  |
| TYPE       | The type of stress and strain defined in TABLED1. (Character; Default = TRUE)  |                                  |
|            | ENG  | Engineering stress and strain.   |
|            | TRUE   | True stress and strain.          |
|            | PLAST  | True stress and plastic strain.  |
|            | PMOD   | Plastic modulus and true stress. |
| TABY       | Number of TABLED1 entry giving the variation of the scale factor for the yield stress (y-value) with the strain rate (x-value). Strain-rate effects are also specified using the Cowper-Symonds relation (see input parameters D and P). See Remark 6. (Integer > 0) |                                  |
| D          | Factor D in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real $\geq 0$ )   |                                  |
| P          | Factor P in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real $\geq 0$ )   |                                  |

### Remarks:

1. A bilinear stress-strain characteristic is used by specifying YIELD and EH:



where the yield stress  $\sigma_y$  is given by

$$\sigma_y = \sigma_0 + \frac{EE_h}{E - E_h} \varepsilon_p$$

where

|                 |   |
|-----------------|---|
| $\sigma_0$      | = yield stress specified in the YIELD field   |
| $E$             | = Young's modulus                             |
| $E_h$           | = hardening modulus specified in the EH field |
| $\varepsilon_p$ | = equivalent plastic strain                   |
| $\sigma_y$      | = yield stress                                |

2. A piecewise linear, stress-strain characteristic is used by specifying TABLE and TYPE (beams and shells only)

$$\sigma_{ij} = [(\sigma_i - \sigma_{i-1})(\varepsilon - \varepsilon_{i-1}) / (\varepsilon_i - \varepsilon_{i-1})] + \sigma_{i+1}$$

The stress-strain characteristic used internally in the solver is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

True stress/true strain (TYPE = TRUE)

Engineering stress/engineering strain (TYPE = ENG)

True stress/plastic strain (TYPE = PLAST)

Plastic modulus/true stress (TYPE = PMOD)

3. With Lagrangian and Eulerian solid elements, only an elastic perfectly plastic yield model is currently used. Only the YIELD field is used.
4. If TABLE is blank or zero, a bilinear stress-strain curve is assumed. If TABLE has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If TABLE is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If TABY is blank or zero and D and P are blank or zero, the yield stress does not vary with strain rate. If TABY has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. (D and P must be blank or zero.)  
If TABY is blank or zero and D and P are defined, the enhancement of the yield stress with strain rate is calculated as

$$\frac{\sigma_d}{\sigma_y} = 1 + \left( \frac{\dot{\varepsilon}_p}{D} \right)^{1/p}$$

Where  $\sigma_d$  is the dynamic stress,  $\sigma_y$  is the static yield stress (YIELD), and  $\dot{\varepsilon}_p$  is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined it will override the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see TABLE and TABY fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. YID must unique among all YLDxx entries in one model.

**YLDZA****Zerilli-Armstrong Yield Model**

Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

Format:

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8    | 9 | 10 |
|-------|-----|---|---|---|---|---|------|---|----|
| YLDZA | YID | A | B | n | C | m | EPS0 |   |    |
|       | D   |   |   |   |   |   |      |   |    |

Example:

|       |     |        |       |     |  |  |  |  |  |
|-------|-----|--------|-------|-----|--|--|--|--|--|
| YLDZA | 7   | 200.E6 | 50.E6 | 0.1 |  |  |  |  |  |
|       | 0.0 |        |       |     |  |  |  |  |  |

| Descriptor | Meaning  |
|------------|--|
| YID        | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A          | Static yield parameters. (Real > 0; Required)                                      |
| B          | Hardening parameters. (Real $\geq 0$ ; Default = 0.0)                              |
| n          | Hardening exponent. (Real; Default = 0.0)  |
| C          | Strain rate parameter. (Real; Default = 1.0)                                       |
| m          | Temperature exponent. (Real; Default = 1.0)  |
| EPS0       | Reference strain rate. (Real > 0; Default = 1.0)                                   |
| CP         | Specific heat. (Real > 0; Default = 1.E20)   |
| D          | Bcc parameter. See Remark 4. (Real; Default = blank)                               |

Remarks:

1. This material model can be used to model Fcc (iron and steels) and Bcc (aluminum and alloys) metals.
2. The yield stress is computed from  
for Fcc metals:

$$\sigma_y = (A + B\dot{\varepsilon}_p^n)\varepsilon^{\left[-mT + CT\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} \text{ for Fcc metals}$$

$$\sigma_y = (A + B\dot{\varepsilon}_p^n) + De^{\left[-mT + CT\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} \text{ for Bcc metals}$$

where

$\varepsilon_p$  = effective plastic strain

$\dot{\varepsilon}$  = effective strain rate

$\dot{\varepsilon}_0$  = reference strain rate

$T$  = temperature

$A, B, n, C, m$ , and  $D$  are constants.

3. The reference strain rate are per unit time.
4. In case the Bcc parameter  $D$  is not supplied, it is assumed that a Fcc metal is defined.

# A

## Configuring the Runtime Environment

- Specifying Parameters
- User-Defined Keywords
- Resolving Duplicate Parameter Specifications
- Customizing Command Initialization and Runtime Configuration Files
- Symbolic Substitution

## Specifying Parameters

MSC Nastran execution is controlled by a variety of parameters, either keywords or special Nastran statements, both required and optional. The purpose of this section is to describe how and where these parameters may be specified, not to describe these parameters in detail. This is done in subsequent sections. The MSC Nastran parameters may be specified on the command line, in a command initialization (INI) file, in runtime configuration (RC) files and, for some parameters, from environment variables. The information from these sources is consolidated at execution time into a single set of values. Much of this information is passed to analysis processing in a "control file", built using the templates ([Customizing the Templates, 51](#)). (The records in this control file are echoed to the .log file.) Examples of INI and RC files are given in the [User-Defined Keywords, 3355](#) and [Customizing Command Initialization and Runtime Configuration Files, 3363](#).

## Command Initialization and Runtime Configuration Files

Although the purposes of the INI and RC files are somewhat different, the format of each file is the same. All INI and RC files are processed twice, once (the "first" pass) to extract parameters (keywords and other information) that are to be used for all MSC Nastran jobs, and once (the "second" pass) to extract parameters specific to a particular job. This is accomplished by separating the INI and RC files into a series of "sections" identified by a "section header" and "subsections" within sections, identified by a subsection "header." There are two types of sections: "unconditional" and "conditional." Subsections are always "conditional."

- An unconditional section is one that starts with the name of the section enclosed in square brackets ("[ ", "]"). Section names may not contain any embedded blanks but may be separated from the square brackets by any number of blanks. As currently implemented, there are three valid unconditional names: "General", "Solver" and "Nastran". (These section names are case-insensitive.) In addition, there is an implicit "unnamed" section that consists of all parameters in the INI or RC file that appear before the first named section or subsection. There is no special meaning assigned to any of the unconditional sections. Their use is optional; the section names are intended to be used for descriptive purposes.
- A conditional section or subsection is one that starts with an expression in the form:

`<keyword><operator><value>`

enclosed in section header identification characters. For a conditional section, the section header identification characters are square brackets ("[ ", "]"), just as for unconditional sections. For a subsection, the section header identification characters are "less than" and "greater than" ("<", ">") characters. Keywords and values may not contain any embedded blanks but may be separated from each other and from the enclosing section header identification characters (the square brackets or "less than"- "greater than" characters) by any number of blanks. In the expression:

- |                               |  |
|-------------------------------|--|
| <code>&lt;keyword&gt;</code>  | represents any valid internal keyword (see <a href="#">Keywords, 182</a> ) or user-defined keyword (see <a href="#">User-Defined Keywords, 3355</a> ). |
| <code>&lt;operator&gt;</code> | specifies the comparison to be performed between <code>&lt;keyword&gt;</code> and <code>&lt;value&gt;</code> as follows:                               |
|                               | =      equal (either string or numeric)  |

|         |   |
|---------|---|
| !       | not equal (either string or numeric)                                  |
| !=      | not equal (either string or numeric)                                  |
| <       | numerically less than   |
| <=      | numerically less than or equal  |
| >       | numerically greater than  |
| >=      | numerically greater than or equal                                     |
| <value> | specifies the appropriate keyword value to be used in the comparison. |
|         | Keywords and values may be specified in any case.                     |

Parameters in unconditional sections, but not in subsections (which are always conditional) within unconditional sections, are processed on the first pass through an INI or RC file. On the second pass, these parameters are ignored (they are not reprocessed). Parameters in conditional sections and subsections are ignored on the first pass. Parameters in conditional sections and subsections whose expressions evaluate to "true" are processed on the second pass through an INI or RC file, thus allowing conditional expressions to reference all of the valid keywords. Note that for subsections within conditional sections, both the conditional expression for the section *and* the conditional expression for the subsection must evaluate to "true" before parameters in the subsection are processed.

Parameter specifications in, either unconditional or conditional sections, may be continued, if necessary, by specifying a backslash ("\") character as the last non-blank character of the line. Note for Windows users, if the parameter value itself ends with a backslash, the statement must have additional characters, such as a comment, after the value specification. For example, a specification such as:

```
sdir=e:\
```

will not work properly. Instead, write the statement as:

```
sdir=e:\ $ Specify the scratch directory
```

In addition to parameters, INI and RC files may contain "comment" records. There are two types of comment records: ignored and printed.

- Ignored comments are records that start with a semi-colon (";") or pound sign ("#"). These records are completely ignored. When running in Windows, there is a special form of ignored comments that may be specified in an INI file (but not in RC files). These are records that start with "REM", short for "REMARK". The test for "REM" is case-insensitive.
- Printed comments are records that start with the currency symbol ("\$"). These records are passed on as part of the analysis information but are otherwise ignored.

**Note:** Although sectioning within INI and RC files was first introduced in MSC Nastran 2004, valid INI and RC files from prior versions of MSC Nastran are fully compatible with this new format. Since sections were not supported in previous versions (except for INI files on Windows, which allowed unconditional sections), all parameters will be in the "unnamed" implicit section (or, on Windows, in named unconditional sections) and will be processed on the first pass through the file. No information will be extracted from these files on the second pass.

The list below specifies the INI and RC files that MSC Nastran uses. Any or all of these files may be omitted. [Table 1-1](#) lists the keywords that are generally set in the unconditional sections of the command initialization file. [Table 1-2](#) lists the keywords that are generally set in RC files.

#### ■ Command Initialization (INI) File

This file is used to define keywords that are to be set whenever the nastran command is executed. Typical keywords in the unconditional sections include the installation base directory and the version of MSC Nastran. Conditional sections and subsections might include keywords such as "rcmd" and "rsdirectory" in sections that are conditional upon the value of the "node" keyword.

LINUX: *install\_dir/prod\_ver/arch/nastran.ini*

Windows: *install\_dir\prod\_ver\arch\nastran.ini*

Starting with MSC Nastran 2011, there are two possible RC files that may be defined in each of the locations that are searched for RC files. The first name is a version independent name and the second name is a version dependent name, where the version number is indicated by *<vernum>* in the file name and the version number for MSC Nastran is 20200.

#### ■ System RC Files

These files are used to define parameters that are applied to all MSC Nastran jobs using this installation structure. Many of the parameters that might be specified in the INI file could, alternatively, be specified in this file.

LINUX: *install\_dir/conf/nastranrc* and

*install\_dir/conf/nast<vernum>.rc*

Windows: *install\_dir\conf\nastran.rcf* and

*install\_dir\conf\nast<vernum>.rcf*

#### ■ Architecture RC Files

This files are used to define parameters that are applied to MSC Nastran jobs using this architecture.

LINUX: *install\_dir/conf/arch/nastranrc* and

*install\_dir/conf/arch/nast<vernum>.rc*

Windows: *install\_dir\conf\arch\nastran.rcf* and  
*install\_dir\conf\arch\nast<vernum>.rcf*

#### ■ Node RC Files

These files are used to define parameters that are applied to MSC Nastran jobs running on this node. Alternatively, the parameters in this file could be specified in a conditional section in one of the previous files, using *nodename* as the value of the "s.hostname" keyword in the conditional expression.

LINUX: *install\_dir\conf\net\nodename\nastranrc* and  
*install\_dir\conf\net\nodename\nast<vernum>rc*

Windows: *install\_dir\conf\net\nodename\nastran.rcf* and  
*install\_dir\conf\net\nodename\nast<vernum>.rcf*

#### ■ User RC Files

These files are used to define parameters that are applied to MSC Nastran jobs run by an individual user.

LINUX: \$HOME/.nastranrc and  
\$HOME/.nast<vernum>rc

Windows: %HOMEDRIVE%&%HOMEPATH%\nastran.rcf and  
%HOMEDRIVE%&%HOMEPATH%\nnast<vernum>.rcf

#### ■ Local RC Files

These files should be used to define parameters that are applied to MSC Nastran jobs that reside in the input data file's directory. This RC file is in the same directory as the input data file. If the "rcf" keyword ([page 214](#)) is used, this local file is ignored.

LINUX: .nastranrc and  
.nast<vernum>rc

Windows: nastran.rcf and  
nast<vernum>.rcf

Please note that the LINUX shorthand "~", to refer to your or another user's home directory, cannot be used in an RC file. In addition, environment variables are only recognized within the context of a logical symbol definition.

Also, note that, on LINUX systems, the leading period (".") on the User RC Files and Local RC Files file names cannot be deleted even if alternate names are specified using the "a.rc" and "a.urcb" keywords as described below.

The file names listed above may be changed by the user using the "a.rc", "a.rcb", "a.urc" and "a.urcb" keywords, noting that the directories in which the files are located may not be changed.

- The “a.rc” keyword can be used to change the names of the version dependent RC file names for the System RC Files, the Architecture RC Files and the Note RC File. The default for this keyword is “nast<vernum>rc” for LINUX and “nast<vernum>.rcf” for Windows.
- The “a.rcb” keyword can be used to change the names of the version-independent RC file names for the System RC Files, the Architecture RC Files and the Node RC Files. The default for this keyword is “nastranrc” for LINUX and “nastran.rcf” for Windows.
- The “a.urc” keyword can be used to change the names of the version dependent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rc” keyword with a leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rc” keyword.
- The “a.urcb” keyword can be used to change the names of the version-independent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rcb” keyword with the leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rcb” keyword.

In addition to keyword specifications, the following MSC Nastran statements (from the NASTRAN and FMS Sections) may appear in RC files and conditional sections in an INI file: NASTRAN, ACQUIRE, ASSIGN, CONNECT, DBCLEAN, DBDICT, DBFIX, DBLOAD, DBLOCATE, DBSETDEL, DBUNLOAD, DBUPDATE, DEFINE, ECHOOFF, ECHOON, ENDJOB, EXPAND, INCLUDE, INIT, PROJ, RESTART and RFINCLUDE. Except for minimal checking of the NASTRAN and PARAM statements, the syntax of these statements is not validated. These records are simply passed on for use in MSC Nastran analysis processing.

INI files and RC files also may contain PARAM statements that specify values that affect MSC Nastran analysis processing. The values associated with PARAM names may be specified using PARAM statements in INI files and RC files or by using PARAM keywords, defined using the PARAM keywords feature as described in [User-Defined Keywords, 3355](#). PARAM statements must be specified in "free-field format", i.e., in the Case Control PARAM format (PARAM,name,value), not in Bulk Data fixed-field format. Please see [Parameters \(Ch. 6\)](#) in the *MSC Nastran Quick Reference Guide* for more information on PARAM names and statements and their usage.

## Environment Variables

Several keywords may have their values set from associated environment variables. When this is the case, the environment variable takes precedence over any INI or RC file keyword specification. A command-line specification will over-ride the environment variable specified value. This same precedence rule applies to user-defined keywords that may have their initial values taken from environment variables, as described in the next section. A list of the keywords and their associated environment variables, along with a description of each keyword, may be obtained by using the following command:

`prod_ver nastran help env`

## User-Defined Keywords

In addition to the internally defined keywords (see [Keywords, 182](#)), MSC Nastran allows users to define their own keywords. There are two classes of user-defined keywords:

- General keywords. These are intended for use in INI file or RC file conditional section clauses, in user modifications to the run template files (nastran.dmp, nastran.lcl, nastran.rmt or nastran.srv) and, for LINUX, in customized queue commands ("submit" keyword).
- PARAM keywords. These are keywords associated with a PARAM name. Using descriptive keywords to set a PARAM value may be more convenient than specifying the PARAM statement in an RC file. Also, keywords are not limited to a maximum of eight characters, as PARAM names are, and may be more descriptive of the action being affected or requested.

User-defined keywords are supported by the "help" and "whence" functions.

## General Keywords

These keywords are defined in the file specified by the "0.kwds" keyword. While the file is not delivered, an administrator could create the file. The default file names are:

LINUX:           *install\_dir/prod\_ver/arch/nastran.kwds*  
Windows:         *install\_dir\prod\_ver\arch\nastran.kwds* or  
                  *install\_dir\bin\nast20200.kwds*  
The file used is the *first* one found.

The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';' or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword records. These records consist of the keyword name along with an optional value descriptor and comment in the form:

```
keyword_name[, attributes] : value_descriptor comment
```

where:

|                               |  |
|-------------------------------|--|
| <code>keyword_name</code>     | is the name to be assigned to the user keyword. This name may not contain any embedded blanks and may not be the same as any internal keyword or previously specified user-defined keyword. It is also case-insensitive except in the case when its initial value may be set from an environment variable with the same name.  |
| <code>attributes</code>       | specifies optional attributes to be assigned to the keyword defined by <code>keyword-name</code> . Currently, the only defined attribute is:<br><code>argv</code> keyword and its value is to be added to the “ <code>r.argv</code> ” keyword value<br>Any number of blanks may separate <code>keyword_name</code> , the separating command and the attributes specification.                      |
| <code>value_descriptor</code> | is optional. If specified, it should be as described in <a href="#">Value Descriptors, 3357</a> and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted.. The default value descriptor is “ <code>string</code> ”. This field may also specify that the initial value of this keyword be taken from an environment variable with the same name. |
| <code>comment</code>          | is an optional comment field. If present, it must be separated from <code>value_descriptor</code> or <code>keyword_name</code> by blanks or must begin with a comment character.   |

There may be any number of leading blanks in the record and before and after the separating colon.

General keywords and the values assigned to them only affect MSC Nastran processing if:

- there are customized INI and RC files that have conditional sections, using these keywords in expressions, that specify other keywords and statements (e.g., NASTRAN and PARAM statements) that modify MSC Nastran processing to meet the requirements of a user's site and installation.
- they are used in customized templates ([Customizing the Templates, 51](#)).
- for LINUX systems, they are used in customized queue commands defined using the "submit" keyword ([Customizing Queue Commands \(LINUX\), 49](#)).

## PARAM Keywords

These keywords are defined in the file specified by the "0.params" keyword. The default file names are:

LINUX:           `install_dir\prod_ver\arch\nastran.params`

Windows:       `install_dir\prod_ver\arch\nastran.params`

The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';' or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword-name records. These records consist of the keyword name, the associated PARAM name, along with an optional value descriptor and comment in the form:

keyword\_name : param\_name : value\_descriptor comment

where:

|                  |  |
|------------------|--|
| keyword_name     | is the name to be assigned to the PARAM keyword. This name is case-insensitive, may not contain any embedded blanks and may not be the same as any internal keyword, general user-defined keyword or previously specified PARAM keyword.   |
| param_name       | is the PARAM name to be associated with keyword_name. This name is case-insensitive, may be a maximum of eight characters, must begin with an alphabetic character and may not contain any embedded blanks. Also, it may not be the same as any previously specified PARAM name. |
| value_descriptor | is optional. If specified, it should be as described in <a href="#">Value Descriptors</a> and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted. The default value descriptor is "string".                                  |
| comment          | is an optional comment field. If present, it must be separated from value_descriptor or param_name by blanks or must begin with a comment character.   |

There may be any number of leading blanks in the record and before and after the separating colons.

Keyword names that are the same as PARAM names are allowed, as long as the keyword name is not an internal or general user-defined keyword name.

Values associated with PARAM names, whether set using PARAM keywords or set using PARAM statements (statements having the form PARAM,*name,value*), directly affect MSC Nastran analysis processing.

## Value Descriptors

Value descriptors enable limited syntax checking for values assigned to general and PARAM user-defined keywords. For general keywords, they may also specify that the initial value of the keyword be set from the value associated with the environment variable having the same name as the keyword. There are two types of syntax checking available: value must be one of a list of entries or value must be numeric. Also, the two forms can be combined. These are specified as follows:

List: { "val1", "val2", ..., "valn" }

That is, the acceptable values are enclosed in double quotes ("") and separated from each other by commas. The specification, including the various acceptable values, may not contain any embedded blanks. Values are case-insensitive and any partial specification is acceptable and will be replaced by the full value. For example, if a keyword may only have the values "preliminary", "check" and "final", the value descriptor would be:

```
{"Preliminary","Check","final"}
```

and a value specification of "Ch" would be accepted and replaced by "check".

Numeric: number

Values will be checked to see if they are valid numbers, either integer or floating point. For example, valid keyword value specifications could be: "1", "-3.247", "4.e-5", "3.75.4", "4.24x" and "-4-5" are invalid specifications.

**Note:** This checking does *not* support the NASTRAN "nnnseee" numeric format, where the 'e' between the number and the signed exponent ("seee") is missing.

Complex value: number, number

This format is only supported for PARAM keyword value descriptors. Values will be checked to see if they consist of two valid numeric values, separated by a comma.

Combined: {"val1", "val2", ..., "valn", number}

**Note:** This "combined" format does not support complex numbers.

In addition, for general keywords, if the value descriptor starts or ends with the string "env", specified in any case and separated from the rest of the value descriptor with a comma (unless the value descriptor is only "env"), the keyword value will be set using the value associated with the environment variable having the same name as the keyword. The environment value will be subjected to the same syntax-checking rules that an INI file, RC file or command line specification would be, with a warning message generated if syntax checking fails. This occurs even if the keyword is specified on the command line. Note that, for LINUX systems, since environment variable names are case-sensitive, the keyword name must be specified exactly the same as the environment variable name. This is the only time that the keyword name is case-sensitive. For Windows systems, since environment variable names are not case-sensitive, this restriction does not apply. Keyword values set from environment variables over-ride keyword values set in INI or RC files but do not over-ride keyword values set on the command line.

If a value descriptor is omitted or is not one of these formats, no syntax checking will be performed.

## Examples:

1. The following value descriptor would accept a value of "test", "final" or a number:

```
{"Test","Final",Number}
```

Acceptable values would be: `te` (replaced by test), `FIN` (replaced by final), `7`, `14.5`, `3.e-4`, `-5`

2. The following value descriptor would accept only the strings "abc", "def", "ghi" and "glm":

```
{"abc","def","ghi","glm"}
```

Acceptable values would be: `g` (replaced by ghi), `aB` (replaced by abc), `g1` (replaced by glm), `D` (replaced by def)

3. The following value descriptor, only valid for a PARAM keyword, would only accept a complex number specification:

```
number,number
```

Acceptable values would be: `1,2`, `7.54`, `3.14`

4. The following value descriptors, only valid for a general keyword, would accept only the strings "qrs", "test", and "xyz". In addition, the value descriptor requests that the keyword value be set from the environment.

```
env,{"qrs","test","xyz"}
```

or

```
{"qrs","test","xyz"},Env
```

Acceptable values would be: `q` (replaced by qrs), `xY` (replaced by xyz), `T` (replaced by test)

## Resolving Duplicate Parameter Specifications

MSC Nastran processing information is obtained by scanning the various INI and RC files, the system environment, and the Nastran command line in the following order:

1. Nastran command line, first pass. Only "program options", i.e., "-x" options, are processed during this command line scan. For example, this is when the "-i ini\_file\_name" program option is processed.
2. Environment variables, first pass. During this pass, the only keywords whose values are set are those that may only be specified as environment variables. This includes keywords such as HOME (for LINUX), HOMEDRIVE and HOME PATH (for WINDOWS) and PWD.
3. INI file, first pass, if this file exists. During this pass, only unconditional sections are processed. Generally, the only keywords processed in this pass are: 0.kwds, 0.params, accmd, acvalid, rcmd, rsdirectory, sysmsg and version (although rcmd and rsdirectory probably should be in conditional sections scanned during the second pass).
4. Environment variables, second pass. During this pass, only those keywords that may only be set in global sections of the INI file or as environment variables are processed. This includes keywords such as MSC\_ARCH, MSC\_BASE and MSC\_VERSD.
5. Nastran command line, second pass. The only general use keywords processed during this command line scan are: dmparallel, jid, jidpath, jidtype, node, pause, rcf, username, version and whence. The processing of other command line keywords is deferred until later command line scans.

This is the time that the user-defined keyword definition files (for both general use and PARAM keywords), if any, are processed and the keyword specifications defined by these files are added to the keywords tables. The keywords defined in these files may be used just as internal keywords are used. (See [User-Defined Keywords, 3355](#).)

6. System RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
7. Architecture RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
8. Node RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
9. User RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
10. Local RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
11. Environment variables, third pass. During this pass, only "general" user-defined keywords that have been flagged to be set from environment variables are processed. (This pass will be skipped if there are no "general" user-defined keywords.)
12. Nastran command line, third pass. Only "general" user-defined keywords are processed during this command line scan. (This pass will be skipped if there are no "general" user-defined keywords.)

At this point, all keyword values that can be used in conditional section expressions are known.

13. INI file, second pass, if this file exists and has conditional sections. During this pass, only the conditional sections are processed.
14. System RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
15. Architecture RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
16. Node RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
17. User RC files, second pass, if these files exists and have conditional sections. During this pass, only the conditional sections are processed.
18. Local RC files, second pass, if these files exist and have conditional sections and if they are not ignored. During this pass, only the conditional sections are processed.
19. Environment variables, fourth pass. During this pass, all keywords that may be set from environment variables and that have not been processed previously are now processed.
20. Nastran command line, fourth pass. All keywords not processed during the previous passes are now processed. For example, this is when user-defined PARAM keyword specifications are processed.  
At this point, all information necessary to generate the "control file" has been collected. This file is generated when the "script templates" (see [Customizing the Templates, 51](#)) are processed.
21. NASTRAN, FMS and PARAM statements in the input file.

If duplicate keywords are encountered, the *last* specification found is the one used. That is, the above list specifies the precedence order, from lowest precedence (number 1) to highest (number 21). The only case in which the last keyword specification is not used is when keywords are "locked", i.e., when a specification of the form

```
lock=keyword
```

is processed. After this "lock" request is processed, any requests to set *keyword*, whether from INI files, RC files, environment variables or command line arguments, are quietly ignored. That is, processing proceeds as if any *keyword* specifications specified after the "lock=keyword" request do not exist. Once a keyword has been "locked," there is no way to "unlock" it. (Note that it is valid to "lock" the lock keyword itself.)

If duplicate NASTRAN and FMS statements are encountered, they are simply passed on for use in MSC Nastran analysis processing in the order in which they were encountered.

Thus, the general rule for resolution is:

- Information specified in NASTRAN input data files always takes precedence over any other values.
- Command line parameters have the next highest precedence.
- Environment variables associated with keywords and that have non-null values are next.
- RC file parameter specifications are next.
- INI file parameter specifications are last.

Generally, the only exceptions to this precedence ordering are "general" user-defined keyword specifications. The command line values take precedence over values specified in unconditional INI file and RC file sections but have lower precedence than values specified in conditional INI file and RC file sections. Because the primary purpose for general user-defined keywords is for conditional section selection, changing a general user-defined keyword in a conditional section *may* lead to unexpected results. Such specifications should be used with care. Also, because user-defined PARAM keywords on the command line are not processed until the last command line scan, PARAM keywords should not be used in INI file and RC file conditional section expressions since command line specified values will not be in effect when these expressions are evaluated.

Because PARAM values may be specified either using PARAM statements or using PARAM keywords, they require further explanation. PARAM statements and PARAM keywords referring to the same PARAM name are considered equivalent definitions for the PARAM name. As such, the last specification, regardless of whether it was a PARAM statement or a PARAM keyword, is the one that is used to establish the value associated with the PARAM name.

## Customizing Command Initialization and Runtime Configuration Files

**Table 1-1** lists the keywords that are generally set in the unconditional sections of the command initialization file.

Table 1-1      Command Initialization File Keywords

| Keyword         | Purpose  |
|-----------------|--|
| <b>0.kwds</b>   | Alternate name for user-defined keywords definition file.  |
| <b>0.params</b> | Alternate name for PARAM keywords definition file  |
| <b>acct</b>     | Enables job accounting, see <a href="#">Enabling Account ID and Accounting Data, 39</a> .  |
| <b>acvalid</b>  | Activates account ID validation, see <a href="#">Enabling Account ID Validation, 39</a> .  |
| <b>MSC_BASE</b> | Defines the installation base directory. Normally this is defined as an environment variable by the <a href="#"><i>prod_ver</i></a> command. |
| <b>version</b>  | Specifies the default version of MSC Nastran to be run.  |

Most of the command line keywords can be set in any of the RC files. **Table 1-2** lists keywords that are generally set in the system, architecture, or node RC files:

Table 1-2      RC File Keywords

| Keyword              | Preferred RC File | Purpose   |
|----------------------|-------------------|---|
| <b>accmd</b>         | System            | Command line to invoke accounting logger program.   |
| <b>acct</b>          | System            | Enables job accounting.   |
| <b>acvalid</b>       | System            | Enables account ID (acid) validation.   |
| <b>authorize</b>     | System            | Specifies the licensing method.   |
| <b>buffsize</b>      | System            | Set the default buffsize. Suggested values are in <a href="#">Table 4-5</a> .   |
| <b>lock</b>          | Any               | Prevent further changes to a keyword's value.   |
| <b>memory</b>        | Node              | Specifies a default memory allocation   |
| <b>memorymaximum</b> | Node              | Specifies a maximum "memory" request. May be specified as a percentage of RAM e.g.<br><br><code>memorymax=0.5xPhysical</code> |
| <b>ncmd</b>          | Architecture      | Specifies the notify command when "notify=yes" is set.  |
| <b>news</b>          | System            | Controls the display of the news file at the beginning of the .f06 file.  |

Table 1-2 RC File Keywords (continued)

| Keyword            | Preferred RC File | Purpose  |
|--------------------|-------------------|--|
| <b>post</b>        | Architecture      | LINUX: Specifies commands to be run after each job is completed.   |
| <b>pcedelta</b>    | Architecture      | LINUX: Specifies the value that is subtracted from the "CPU" keyword value to determine the NQS per-process CPU time limit.                              |
| <b>ppmdelta</b>    | Architecture      | LINUX: Specifies the value that is added to the "memory" keyword value to determine the NQS per-process memory limit.                                    |
| <b>pre</b>         | Architecture      | LINUX: Specifies commands to be run before each job begins.  |
| <b>prmdelta</b>    | Architecture      | LINUX: Specifies the value that is added to the "ppm" value to determine the NQS per-request (per-job) memory limit.                                     |
| <b>qoption</b>     | Architecture      | LINUX: Specifies a string of additional queuing options to be set in the queue submittal command.  |
| <b>rcmd</b>        | Any               | Specifies the remote Nastran command to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression. |
| <b>real</b>        | Node              | Specifies the "REAL" parameter to limit virtual memory usage.  |
| <b>rsdirectory</b> | Any               | Specifies the scratch directory to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression.      |
| <b>scratch</b>     | Any               | Specifies the default job status as scratch or permanent.  |
| <b>sdirectory</b>  | Node              | Specifies a default scratch directory.   |
| <b>submit</b>      | Architecture      | LINUX: Defines queues and their associated submittal commands.   |
| <b>sys<b>n</b></b> | Any               | Specifies system cells. Can also be specified using the synonym keywords, e.g., bufsize is equivalent to sys1.   |

## Examples

The following (relatively simplistic) examples illustrate how unconditional and conditional sections could be used.

### Example 1:

Assumptions: There are three computer nodes, sysnode1, sysnode2 and sysnode3, that may be accessed.

On sysnode1:

- MSC Nastran 2014 and MSC Nastran 2020 are installed:
  - MSC Nastran 2014 is accessed using "/local/msc/bin/nast2014"
  - MSC Nastran 2020 is accessed using "/local/msc/bin/nast20200"
  - The scratch directory is /local/temp

On sysnode2:

- Only MSC Nastran 2014 is installed and is accessed using "/local1/msc/bin/nast2014"
- The scratch directory is /local1/temp

On sysnode3:

- MSC Nastran 2014 and MSC Nastran 2020 are installed:
  - MSC Nastran 2014 is accessed using "/local2/msc/bin/nast2014"
  - MSC Nastran 2020 is accessed using "/local2/msc/bin/nast20200"
- The scratch directory is /local2/temp

All of this information could be specified in an INI file, identical on all three nodes, as follows:

```
;  
; This is the MSC Nastran Command Initialization File  
; The default version is to be set to 2020.0  
;  
version=2020.0  
; Define conditional sections giving the appropriate sdir  
; values when MSC Nastran is run locally.  
  
[ s.hostname = sysnode1 ]  
sdir=/local/temp  
[ s.hostname = sysnode2 ]  
sdir=/local1/temp  
[ s.hostname = sysnode3 ]  
sdir=/local2/temp  
  
; Define conditional sections giving the appropriate  
; remote access keywords when a "node" value,  
; requesting remote execution, is specified.  
;  
[ node = sysnode1 ]  
rsdir=/local/temp  
< version = 2014.0 >
```

```
rcmd=/local/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local/msc/bin/nast2000

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast2014

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local2/msc/bin/nast20200

;

; This is the end of the Command Initialization file
;
```

Alternatively, the information could be split between an INI file and a system RC file, identical on all three nodes, as follows:

In the INI file:

```
;

; This is the MSC Nastran Command Initialization File
; The default version is to be set to 2020.0
;
version=2020.0

; Define conditional sections giving the appropriate
; remote access keywords when a "node" value,
; requesting remote execution, is specified.
;
[ node = sysnode1 ]
rsdir=/local/temp
< version = 2014.0 >
rcmd=/local/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local/msc/bin/nast20200

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast2014

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2020.0 >
rcmd=/local2/msc/bin/nast20200
```

```
;  
; This is the end of the Command Initialization file;
```

In the system RC file, identical on all three nodes:

```
;  
; This is the MSC Nastran system RC file.  
;  
; Define conditional sections giving the appropriate sdir  
; values when MSC Nastran is run locally.  
  
[ s.hostname = sysnode1 ]  
sdir=/local/temp  
[ s.hostname = sysnode2 ]  
sdir=/local1/temp  
[ s.hostname = sysnode3 ]  
sdir=/local2/temp  
  
;  
; This is the end of the system RC file  
;
```

### Example 2:

Assumptions: User keywords defining "run type" and "data complexity" are needed and AUTOSPC, AUTOSPCR, BAILOUT and ERROR PARAM values are to be set based on these keywords.

The nastran.kwds file could be:

```
; User Keywords  
Runtype:{"prelim","development","final"};Analysis stage  
    Level : number          # Data complexity level  
    ;
```

The nastran.params file could be:

```
; PARAM keywords  
  
Set_AutoSPC : AutoSPC : {"Yes","No"}  
Set_AutoSP_Cr : AUTOSPCR : {"yes","no"}  
Bailout_Value : bailout : number  
Set_Error : Error : number  
;
```

Then, the system RC file could contain:

```
; RC file  
[ runtype = prelim ]  
set_automspc = yes  
bailout_value = -1  
set_error = 0  
set_automspc_cr = yes  
  
[ runtype = development ]  
set_automspc=yes  
bailout_value=0
```



```
set_error=-1

[runtype=final]
set_automspc=no
param,bailout,0
param,error,-1
param,autospcr,no

[level < 3]
; basic data complexity parameters
[level >= 3]
<level>8>
; advanced data complexity parameters

<level<=8>
; intermediate data complexity parameters

; End of RC file
```

## Symbolic Substitution

### Introduction

Symbolic Substitution is a capability added to MSC Nastran that allows a user to effectively modify a Nastran data file using command line and RC file keyword specifications without actually editing the file. This capability is very similar to “environment variable” expansion that happens in various command prompt shells such as the Linux Bourne, Korn and C shells and the Windows Command Prompt shell when scripts are processed. It is also analogous in some ways to the capabilities provided by programming language preprocessors, for example, the CPP preprocessor used by the various C/C++ compilers. The key feature of symbolic substitution is that these modifications do *not* affect the actual data file but present the data read from the data file to the processing program as if it was the modified data that was being processed.

Generally, symbolic substitution means that a data record is scanned to see if it contains special data strings (that identify the “symbolic” variables) that specify “symbolic substitution” requests. If such strings are found, the record is modified to replace the special data strings with user-defined substitution (replacement) strings (the values currently associated with the “symbolic” variables, i.e., the variable “values”) and it is this modified record that is actually processed. This symbolic substitution happens before any other processing of the record occurs, thus making it transparent to the rest of the program processing the data record. In the case of MSC Nastran, this symbolic substitution processing will happen immediately after a record is read from the Nastran data file and before any other processing (with the possible exception of special processing required to satisfy licensing requirements) is performed.

### Simple Examples

Two very simple examples illustrate how this capability could be used in Nastran data files. Note that the details of the syntax are completely described in the following sections and may be ignored for now. Also note that the examples do not deal with things such as managing the output from multiple Nastran runs. These issues, involving, among other techniques, using command line or RC file keywords such as "out=", "append=" and "old=yes", are beyond the scope of this document.

#### Example 1:

Suppose you want to make several tests where the thickness of a PSHELL element is to be varied. You could do this by defining the thickness of the PSHELL element as a “symbolic variable” (identified using the string "%thickness%"), setting a default value (using the "%defrepsym" statement) and specifying the desired thickness on the command line (using the "REPSYM=" keyword). A very simple data file (sym.dat) could be (where most of the BULK entries are in an include file named "model.bdf", not shown here):

```
%defrepsym thickness=5.0
SOL 103
CEND
TITLE = 1st perturbation, t = %thickness%
ECHO = NONE
SUBCASE 1
METHOD = 100
```

```

SPC = 1
DISP = ALL
BEGIN BULK
EIGRL,100,,,6
PARAM,POST,0
PARAM,GRDPNT,0
$PBEAML Properties
PBEAML   2      1      I
          70.0    60.0    60.0    3.3     5.     5.
$
$PSHELL Properties
$
pshell,1,1,%thickness%,1,,1
$
include 'model.bdf'
enddata

```

If the test is run using the following command line:

```
nast20200 sym repsym=thickness=1.0 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 1.0
```

and

```
pshell,1,1,1.0,1,,1
```

If the test is run using the following command line:

```
nast20200 sym repsym=thickness=3.5 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 3.5
```

and

```
pshell,1,1,3.5,1,,1
```

If the test is run without specifying any REPSYM setting for "thickness", e.g., using the following command line:

```
nast20200 sym ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 5.0
```

and

```
pshell,1,1,5.0,1,,1
```

### **Example 2:**

Suppose you have a test that contains two superelements, where the only difference between the data for each superelement is the area of a PBAR element. Instead of having two different definitions, you could have a single definition of the data in an include file, where the area of the PBAR is specified as a symbolic variable. The include file (called "bar.bdf") could be:

```
%defrepsym area=1.  
grid,2,,1.0,0.0,0.0  
grid,3,,2.0,0.0,0.0  
grid,4,,3.0,0.0,0.0,,123456  
cbar,2,2,2,3,0.,1.,0.  
cbar,3,2,3,4,0.,1.,0.  
pbar,2,2,%area%,1.,1.,1.  
mat1,2,1.e7,,,3
```

and the actual input file could be:

```
sol 101  
cend  
title=simple part se  
echo=both  
subcase 1  
load=1  
disp=all  
elforce=all  
begin bulk  
grid,1,,0.0,0.0,0.0  
grid,2,,1.0,0.0,0.0  
cbar,1,1,1,2,0.,1.,0.  
pbar,1,1,1.,1.,1.,1.  
mat1,1,1.e7,,,3  
force,1,1,,1.,1.,1.,1.  
$  
begin super=1  
%setrepsym area=1.  
include 'bar.bdf'  
$  
begin super=2  
%setrepsym area=2.  
include 'bar.bdf'  
enddata
```

The first "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,1.,1.,1.,1.
```

and the second "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,2.,1.,1.,1.
```

## Detailed Specifications

The use of the Symbolic Substitution capability is defined by a number of “rules”. These “rules” are given in the following sections and provide the complete specification. Following the rules, there is information about requesting report information and about error handling. Finally, there are some (again simple) examples showing usage.

## Symbolic Substitution Rules

The following rules define the symbolic substitution user interface. The descriptions start with the rules for variable naming, followed by the rules for defining the replacement width information, followed by the various keywords and statements used to control symbolic substitution.

### Variable Naming

The rules for naming symbolic substitution variables are:

- Symbolic variable names are not case-sensitive, are a maximum of 32 characters long and may not contain leading, trailing or embedded blanks or special characters including ("\_"). Variable names must start with an alphabetic character followed by zero or more alphabetic or numeric characters. For example:
  - The variable name "VaRiaBLe1" is the same as "VARIABLE1" and "variable1"
  - The following variable names are valid:
    - abcdef
    - abc123
    - Name1
  - The following variable names are not valid:
    - 123abc      Does not start with an alphabetic character
    - a bcd      Contains an embedded blank
    - abc&      Contains an invalid character ('&')
    - /def      Does not start with an alphabetic character
    - \_abc123      Uses an underscore in the name.
- Unless symbolic variable values are quoted, they are not case-sensitive and may not contain leading, trailing or embedded blanks or percent ('%') characters. The quoting rules are given below.

### Substitution Field Width Specification

The ability to control the appearance of any symbolic substitution is an important requirement when generating data for a program such as MSC Nastran. The result of a symbolic substitution request is identified as a *field*. Substitution field width information can be taken by default, specified in the data file or specified using command line and/or RC file keywords. These methods are explained below.

The rules for defining substitution field width information are:

- Symbolic variable substitution is, by default, *exact*. That is, the number of characters occupied by the symbolic symbol replacement is exactly the same as the replacement value. However, this default replacement processing can be controlled by specifying the substituted field *width*, the field *precision* and the *justification* within the field. This information is specified using the syntax

-w.p

where the ‘-‘, ‘w’ and ‘p’ are all optional and have the following meanings.

- The field width specification (w) defines the *minimum* number of characters the field is to have as a decimal integer value. If the replacement value has fewer characters than the field width, it will be padded with spaces on the left (by default) or on the right (if the left justification flag is specified). If the replacement value has more characters than the field width and if no precision value was specified, the entire replacement string will be used. A field width value of 0 (zero) is equivalent to omitting the width specification. Note that a negative width value will be processed as if the “left-justification” flag was specified (see below) since a negative field width is meaningless.
  - The field precision specification (p) defines the *maximum* number of characters the field is to have. The format is a period (.) followed by a decimal integer value. If the replacement value length exceeds the precision value, only the last p (by default) or the first p (if the left justification flag is set) characters of the replacement value will be used. A field precision value of 0 (zero) (or a negative value) is equivalent to omitting the precision specification.
  - If both field width and field precision are specified and are positive, the precision value cannot be less than the width value. If it is, it will be reset to the field width.
  - The ‘-‘ character is the “left-justification” flag and specifies that the replacement value is to be left-justified within the field. If this character is omitted, the replacement value will be right-justified within the field.
- For example, the width, precision and justification of a typical field in the Bulk Data portion of a Nastran data file is:

-8 . 8

meaning that the field is exactly eight characters wide and that data is to be left-justified within the field. For a wide-format Bulk Data record, this specification would be:

-16 . 16

The specification for an exact replacement, i.e., where the replaced field is exactly the size of the replacement value, is:

0 . 0

- To simplify width specification for Nastran widths, the following (case-insensitive) synonyms for common widths are available and may be used wherever a width specification can be used:

|       |                           |
|-------|---------------------------|
| exact | is equivalent to 0 . 0    |
| bulk  | is equivalent to -8 . 8   |
| wide  | is equivalent to -16 . 16 |

It is very important to note that there are two distinct portions to a Nastran data file, that part that is before the first BEGIN statement and that has “free format”, and that part that is after the first BEGIN statement (the Bulk Data Section) and often has fixed format fields. Because of this, two different sets of field width information are maintained for use when field width information is not explicitly specified as part of a symbolic substitution request, one for use before the first BEGIN statement and one for use after the first BEGIN statement.

## Defining Variable Values and Width Information

Symbol names and associated values and symbol width specifications may be set using keywords on the command line or in RC files and may be set using special statements in the Nastran data file itself. Each keyword and statement is explained in detail.

### *Using Command Line or RC File Keywords*

#### Setting Variable Value Using REPSYM

Symbolic variables and associated values may be set on the Nastran command line or in RC files using the keyword

```
repsym=<varname>=<varvalue>
```

where <varname> specifies the name of the symbolic variable and <varvalue> specifies the value to be associated with the variable name. For example,

```
repsym=abc=1.23e-5
```

#### Setting Variable Width Information Using REPWIDTH

Symbolic variable substitution default width information may be set on the Nastran command line or in RC files using the keyword

```
repwidth=<widthinfo1>,<widthinfo2>
```

where <widthinfo1> specifies the default width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described previously. If either width specification is omitted, the current default for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
repwidth=12,bulk
```

specifies that symbolic substitution default width is to be 12.0 before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered and

```
repwidth=,bulk
```

specifies that symbolic substitution default width is to be EXACT (or 0.0, the default) before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered.

Just as with other Nastran command line or RC file keywords, the REPSYM and REPWIDTH keywords are not case-sensitive.

### *Using Special Statements in a Nastran Data File*

#### Setting Values Using setrepsym

Symbolic variables and associated values may be set in a Nastran data file using the following statement:

```
%setrepsym <varname>=<varvalue>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following <varvalue>, where the start of the comment is indicated by a ' \$' (blank,

currency symbol). The `%setrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example,

```
%setrepsym abc=1.23e-5
```

### Clearing ("Unsetting") Values Using `unsetrepsym`

A symbolic variable value set using the `%setrepsym` statement may cleared ("unset") in a Nastran data file using the following statement:

```
%unsetrepsym <varname>
```

where the '`%`' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a '`$`'. The `unsetrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the variable `abc`, use

```
%unsetrepsym abc
```

### Setting Default Values Using `defrepsym`

Default variable values can be set in a Nastran data file using the following statement:

```
%defrepsym <varname>=<varvalue>
```

where the '`%`' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varvalue>`, where the start of the comment is indicated by a '`$`'. The `defrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. The specified value will be used *only* if a value for `<varname>` was not previously set, i.e., by a `repsym` keyword on the command line or in an RC file or by a `%setrepsym` statement previously specified in the data file that has not been unset by a `%unsetrepsym` statement. For example,

```
%defrepsym abc=2.46e+2
```

### Clearing ("Unsetting") Default Values Using `undefrepsym`

The default value for a symbolic variable may cleared ("unset") in a Nastran data file using the following statement:

```
%undefrepsym <varname>
```

where the '`%`' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a '`$`'. The `undefrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the default value associated with variable `abc`, use

```
%undefrepsym abc
```

### Setting Width Information Using `setrepwidth`

Symbolic variable substitution default width information may be set in a Nastran data file using the following statement:

```
%setrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following <widthinfo2>, where the start of the comment is indicated by a ' \$'. The setrepwidth string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. <widthinfo1> specifies the width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
%setrepwidth 0.0,wide
```

specifies that the symbolic substitution width specification is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

#### Clearing ("Unsetting") Width Information Using unsetrepwidth

Symbolic variable substitution width information set using the %setrepwidth statement may be cleared in a Nastran data file using the following statement:

```
%unsetrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the unsetrepwidth string, where the start of the comment is indicated by a ' \$. The unsetrepwidth string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both width specifications.

#### Setting Default Width Information Using defrepwidth

Default symbolic variable substitution width information may be set in a Nastran data file using the following statement:

```
%defrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record (except for optional comments following <widthinfo2>, where the start of the comment is indicated by a ' \$. The defrepwidth string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. <widthinfo1> specifies the default width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a -w.p specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
%defrepwidth 0.0,wide
```

specifies that default symbolic substitution is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

#### Clearing ("Unsetting") Default Width Information Using undefrepwidth

Default symbolic variable substitution width information may be cleared in a Nastran data file using the following statement:

```
%undefrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the `undefrepwidth` string, where the start of the comment is indicated by a ' \$'. The `undefrepwidth` string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both default width specifications.

### ***General Information For Special Statements***

The `%setrepsym`, `%unsetrepsym`, `%defrepsym`, `%undefrepsym`, `%setrepwidth`, `%unsetrepwidth`, `%defrepwidth` and `%undefrepwidth` statements are deleted, logically, from the data file and will never be processed by the rest of Nastran unless an error is encountered while they are being processed. This is discussed in the [Error Handling, 3380](#).

### **Requesting Symbolic Substitution**

Symbolic variable substitution will occur when a string having the form

```
%<varname>,<widthinfo>:<varvalue>%
```

is found anywhere within a Nastran data file, except that this string may *not* span records, i.e., the substitution request must be on a single record (line). The leading and trailing '%' characters are required as is the `<varname>` field. The `<widthinfo>` field is optional. If it is omitted, the comma (,) separating it from the `<varname>` field may be omitted and the rules for determining what width specification will be used are discussed below. The `<varvalue>` field is optional and provides a way of specifying a default value, i.e., the "local default value", as described below. If it is omitted, the colon (:) separating it from the `<varname>` (or `<widthinfo>`) field may be omitted. The rules for determining what symbolic value will be used as the substitution value are discussed below. For example, if the symbolic variable abc is to be replaced by its current value with no special processing (or if default width processing is to be used), the substitution request would be:

```
%abc%
```

If the symbolic variable is to be replaced by its current value, with the minimum field width to be 12 characters and with the value always to be left-justified, the substitution request would be:

```
%abc,-12%
```

### **Quoting Rules For Symbolic Variable Values**

- If a symbolic variable value is case-sensitive, if it contains leading, trailing or embedded blanks or if it contains percent characters, tab characters or other special characters, it must be quoted. (Note that "escape" sequences such as '\t' or '\n' are not given any special treatment; that is, they are left as is.)
  - If the value is part of a `repsym` keyword command-line specification, the quoting rules of the command shell being used apply.
  - If the value is part of a `repsym` keyword specified in an RC file, it must be enclosed in single quotes (').

- If the value is part of a %setrepsym or %defrepsym record or if it specified as the “local default value” in a symbolic substitution request, quoting a symbolic variable value means enclosing the value in one of the following pairs of characters:

| Starting Quote Character | Ending Quote Character |
|--------------------------|------------------------|
| "                        | "                      |
| '                        | '                      |
| /                        | /                      |
| \                        | \                      |
| [                        | ]                      |
| {                        | }                      |
| (                        | )                      |

If the first non-blank character encountered in a variable value specification is one of the starting quote characters, the variable value *must* be ended by the associated ending quote character. The actual variable value will be the (possibly null) string between (but not including) the starting and ending quote characters. If the variable value starts with one of the starting quote characters, it must be quoted using an alternate quote character.

### General Rules For Symbolic Variable Substitution

- Nested symbolic substitution is not supported. Even if the value associated with a symbolic variable name is, itself, in the format of a symbolic variable substitution request, that request will be ignored. That is, after symbolic variable substitution has occurred, the substituted string is *not* re-scanned.
- Determining what symbolic variable value will be used when a variable substitution request is encountered depends on where the variable value associated with the specified variable name was set. The *first* value encountered in the following hierarchy is the value that will be used:
  - A value specified in the Nastran data file using the %setrepsym statement, if there is one active, i.e., if it has not been deactivated by a %unsetrepsym statement.
  - A value specified on the Nastran command line or in RC files using the repsym keyword.
  - As part of the variable symbol substitution request, using the local default value, if there is one.
  - A value specified in the Nastran data file using the %defrepsym statement, if there is one active, i.e., if it has not been deactivated by a %undefrepsym statement.

This precedence follows normal MSC Nastran ordering, i.e., "the data file wins," while still providing great flexibility. Also, the ordering of the last two items in this hierarchy allows a user to set all defaults except for special cases and follows the idea that the specification "closest" to the use is the one used. If no replacement value is found, the substitution request will be ignored and the record will be unchanged.

- Determining what symbolic width specification will be used when a variable substitution request is encountered depends on where the width information has been specified and on the part of the Nastran data file that is being processed, i.e., is the variable substitution request before or after the first BEGIN statement. The *first* width specification value encountered in the following hierarchy is the specification that will be used:
  - A value specified in the symbolic substitution request itself, i.e., if a <widthinfo> entry was specified as part of the symbolic substitution request.
  - A value specified on a %setrepwidth statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by an %unsetrepwidth statement.
  - A value specified on the Nastran command line or in RC files using the repwidth keyword corresponding to the current section in the Nastran data file.
  - A value specified in the Nastran data file using the %defrepwidth statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by a %undefrepwidth statement.
  - The program default value of exact (0 . 0).

This precedence also follows normal Nastran ordering, i.e., "the record wins followed by the data file wins," while still providing great flexibility.

- When running in licensing "Interlock" mode, i.e., in CRC validation mode, the following restrictions will be in effect. If a restriction is violated, Nastran processing will be terminated.
  - The %setrepsym, %unsetrepsym, %defrepsym and %undefrepsym statements are not allowed. Also, specifying a default value within the symbolic substitution request is not allowed. That is, symbolic variable values may only be set using the repsym keyword on the command line or in an RC file. Note that the %setrepwidth, %unsetrepwidth, %defrepwidth and %undefrepwidth statements *are* allowed.
  - A maximum of two symbolic substitution specifications are allowed per record and a maximum of ten symbolic substitution requests are allowed in the entire input data file.
  - Interlock CRC calculations will be made on the input record *before* symbolic substitution occurs. Note that any alterations to the record made as part of the CRC calculation processing will not affect symbolic substitution processing.

## Requesting Symbolic Substitution Replacement Information Using REPINFO

- A report of what symbolic substitutions were made is generated at the end of Nastran processing, with the level of detail in the report controlled by an "information level" flag set using the `repinfo=n` keyword, where *n* is an integer number that specifies the level of detail desired. The meanings the various values for *n* are as follows:

- 0 suppress the report altogether
- 1 report the various values assigned using the repsym keyword

- 2 same as 1 except add the various values assigned using the `setrepssym` statement
- 3 same as 2 except add the various values assigned using the `defrepssym` statement
- 4 same as 3 except add the various values assigned as local default values
- 5 same as 1 except add all locations where the specified `repssym` value was used
- 6 same as 2 and 5 except add all locations where the specified `setrepssym` value was used
- 7 same as 3 and 6 except add all locations where the specified `defrepssym` value was used
- 8 same as 4 and 7 except add all locations where local default values were used.

The report is written to the .f06 file. If there is not enough dynamic memory available to save the report information, the `repinfo` level may be reduced. When running in MSC Nastran, the default is `repinfo=1`. Otherwise, `repinfo=0` will be forced.

- Just as with other Nastran command line or RC file keywords, the `REPINFO` keyword is not case-sensitive.

## Error Handling

If an error is encountered processing a `setrepssym`, `unsetrepssym`, `defrepssym`, `undefrepssym`, `setrepwidth`, `unsetrepwidth`, `defrepwidth` or `undefrepwidth` statement, a comment string will be added to the record giving the error information and the record will be passed to Nastran (or the application reading the data file) as if the record was a normal Nastran data record. If an error is encountered in a record containing a symbolic substitution request, the symbolic substitution request will not be processed and, if `repinfo=1` or greater is in effect, a message giving information about the error will be written to the .log file. It is expected that the statements in error will not be valid Nastran statements and so will be flagged as an error.

## Examples

1. The value on an “OPTION” statement is to be settable using the command line, taking a default value of “OPT1val” (case-sensitive) if no command line value is set. The OPTION statement could be

```
OPTION=%Option:'OPT1val'%
```

and the command line parameter that would be used to set OPTION to a different value, OP2VAL (not case-sensitive), would be

```
RepSym=Option=op2val
```

2. An INCLUDE file contains records that are to be used four times in the Bulk Data Section of a Nastran data file, with the only difference being the value in Field 3 of one record. The first time the file is used, this field must contain the value 1.234, the second time this field must contain the value 4.567 and the last two times this field must contain the value -12.578. In all cases, the replacement field must be eight characters wide and the data must be left-justified in the field. Assuming that the symbolic variable is DATFL3 and that the include file name is incl.data, this could be done as follows:

In the include file, specify the following statements before the record to be modified:

```
%DefRepSym datfl3=-12.578
```

then the record to be modified could be specified as follows:

```
FL1      FL2      %datfl3%FL4      FL5      FL6
```

and, for completeness, specify the following record after the record to be modified:

```
%UndefrepSym datfl3
```

Then the data file would contain:

```
....  
%setrepSym DATFL3=1.234
```

```
%DefRepWidth ,bulk
```

```
include 'incl.data'
```

```
....  
%setrepSym DATFL3=4.567
```

```
include 'incl.data'
```

```
%UnsetrepSym datfl3
```

```
....  
include 'incl.data'
```

```
....  
include 'incl.data'
```

